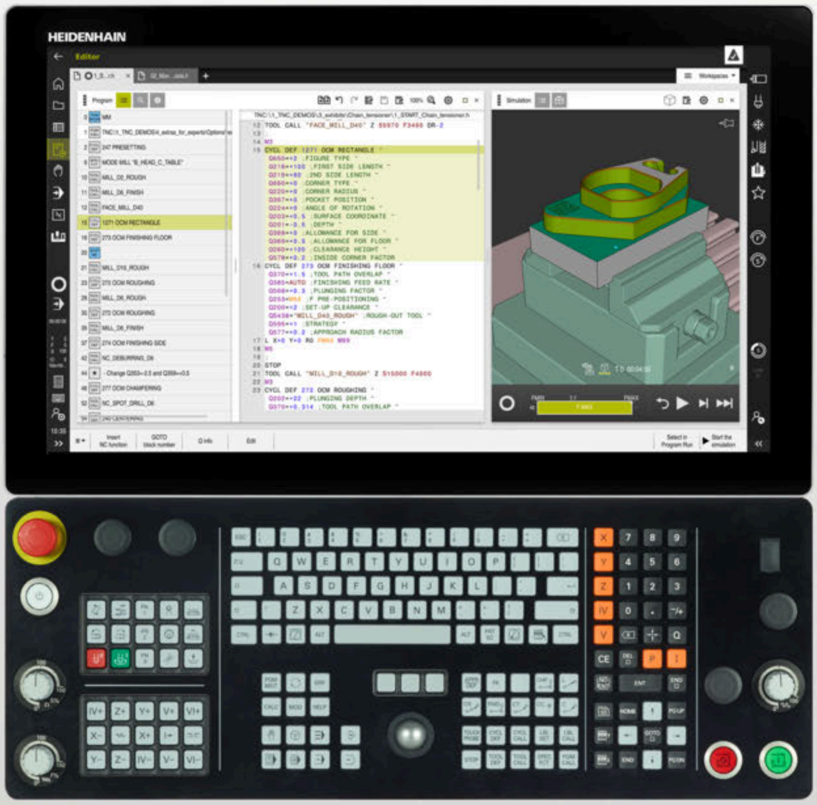
HEIDENHAIN

TNC7

TNCguide

NC Software
817620-16
817621-16
817625-16



English (en)
01/2022

Contents

1

About the User's Manual

1.1 Target group: Users

A user is anyone who uses the control to perform at least one of the following tasks:

- Operating the machine
  - Setting up tools
  - Setting up workpieces
  - Machining workpieces
  - Eliminating possible errors during program run
- Creating and testing NC programs
  - Creating NC programs at the control or externally using a CAM system
  - Using the Simulation mode to test the NC programs
  - Eliminating possible errors during program test

The depth of information in the User's Manual results in the following qualification requirements on the user:

- Basic technical understanding, e.g. ability to read technical drawings and spatial imagination
- Basic knowledge in the field of metal cutting, e.g. meaning of material-specific parameters
- Safety instructions, e.g. possible dangers and their avoidance
- Training on the machine, e.g. axis directions and machine configuration

> HEIDENHAIN offers separate information products for other target groups:
>
> - Leaflets and overview of the product program for potential buyers
> - Service Manual for service technicians
> - Technical Manual for machine manufacturers
>
> Additionally, HEIDENHAIN provides users and lateral entrants with a wide range of training opportunities in the field of NC programming
> **HEIDENHAIN training portal**

In line with the target group, this User's Manual only contains information on the operation and use of the control. The information products for other target groups contain information on further product life phases.

1.2 Available user documentation

User's Manual

HEIDENHAIN refers to this information product as User's Manual, regardless of the output or transport medium. Well-known designations with the same meaning include operator’s manual and operating instructions.

The User's Manual for the control is available in the variants below:

- As a printed version, sub-divided into the modules below:
  - The **Setting up and running** User's Manual contains all information needed for setting up the machine and for running NC programs.
    ID: 1358774-xx
  - The **Programming and testing** User's Manual contains all information needed for creating and testing NC programs. Touch probe and machining cycles are not included.
    ID for Klartext programming: 1358773-xx
  - The **machining cycles** User's Manual contains all functions of the machining cycles.
    ID: 1358775-xx
  - The **Measuring cycles for Workpieces and Tools** User's Manual contains all functions of the touch probe cycles.
    ID: 1358777-xx
- As PDF files, sub-divided according to the printed versions or as complete PDF, containing all modules
  TNCguide
- As HTML file for use as an integrated product help **TNCguide** directly on the control
  TNCguide

The User's Manual supports you in the safe handling of the control according to its intended use.
Further information: "Proper and intended operation", Page 68

Further information products for users

The following information products are available to you as users:

- **Overview of new and modified software functions** informs you about the innovations of specific software versions.
  TNCguide
- **HEIDENHAIN brochures** inform you about products and services by HEIDENHAIN, e.g. software options of the control.
  HEIDENHAIN brochures
- The **NC Solutions** database offers solutions for frequently occurring tasks.
  HEIDENHAIN NC solutions

1.3 Types of notes used

Safety precautions

Comply with all safety precautions indicated in this document and in your machine manufacturer's documentation!

Precautionary statements warn of hazards in handling software and devices and provide information on their prevention. They are classified by hazard severity and divided into the following groups:

⚠ DANGER
Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **will result in death or severe injury.**

⚠ WARNING
Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in death or serious injury**.

⚠ CAUTION
Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in minor or moderate injury.**

NOTICE
Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in property damage**.

Sequence of information in precautionary statements

All precautionary statements contain the following four sections:

- Signal word indicating the hazard severity
- Type and source of hazard
- Consequences of ignoring the hazard, e.g.: "There is danger of collision during subsequent machining operations"
- Escape – hazard prevention measures

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the software.
In these instructions, you will find the following informational notes:

	The information symbol indicates a **tip**. A tip provides additional or supplementary information.

	This symbol prompts you to follow the safety precautions of your machine manufacturer. This symbol also indicates machine-dependent functions. Possible hazards for the operator and the machine are described in the machine manual.

	The book symbol represents a **cross reference** to external documentation, e.g. the documentation of your machine manufacturer or other supplier.

Have you found any errors or would you like to suggest changes?

We continuously strive to improve our documentation for you. Please help us by sending your suggestions to the following e-mail address:

tnc-userdoc@heidenhain.de

1.4 Notes on using NC programs

NC programs contained in this User's Manual are suggestions for solutions. The NC programs or individual NC blocks must be adapted before being used on a machine.

Change the following contents as needed:

- Tools
- Cutting parameters
- Feed rates
- Clearance height or safe position
- Machine-specific positions, e.g. with **M91**
- Paths of program calls

Some NC programs depend on the machine kinematics. Adapt these NC programs to your machine kinematics before the first test run.

In addition, test the NC programs using the simulation before the actual program run.

	With a program test you determine whether the NC program can be used with the available software options, the active machine kinematics and the current machine configuration.

1.5 User's Manual as integrated product help: TNCguide

Application

The integrated product help **TNCguide** offers the full content of all User's Manuals.

Further information: "Available user documentation", Page 59

The User's Manual supports you in the safe handling of the control according to its intended use.

Further information: "Proper and intended operation", Page 68

Requirement

In the factory default setting, the control offers the integrated product help **TNCguide** in German and English language versions.

If the control cannot find a **TNCguide** language version matching the selected dialog language, it opens **TNCguide** in English.

If the control cannot find a **TNCguide** language version, it opens an information page with instructions. With the link available there and the steps provided, you can supplement the files missing in the control.

> You can also open the information page manually by selecting the **index.html** e.g. at **TNC:\tncguide\en\readme**. The path depends on the desired language version, e.g. **en** for English.
>
> With the steps provided you can also update the **TNCguide** version. Updating may be required e.g. after a software update.

Description of function

The integrated **TNCguide** product help can be selected within the **Help** application or in the **Help** workspace.

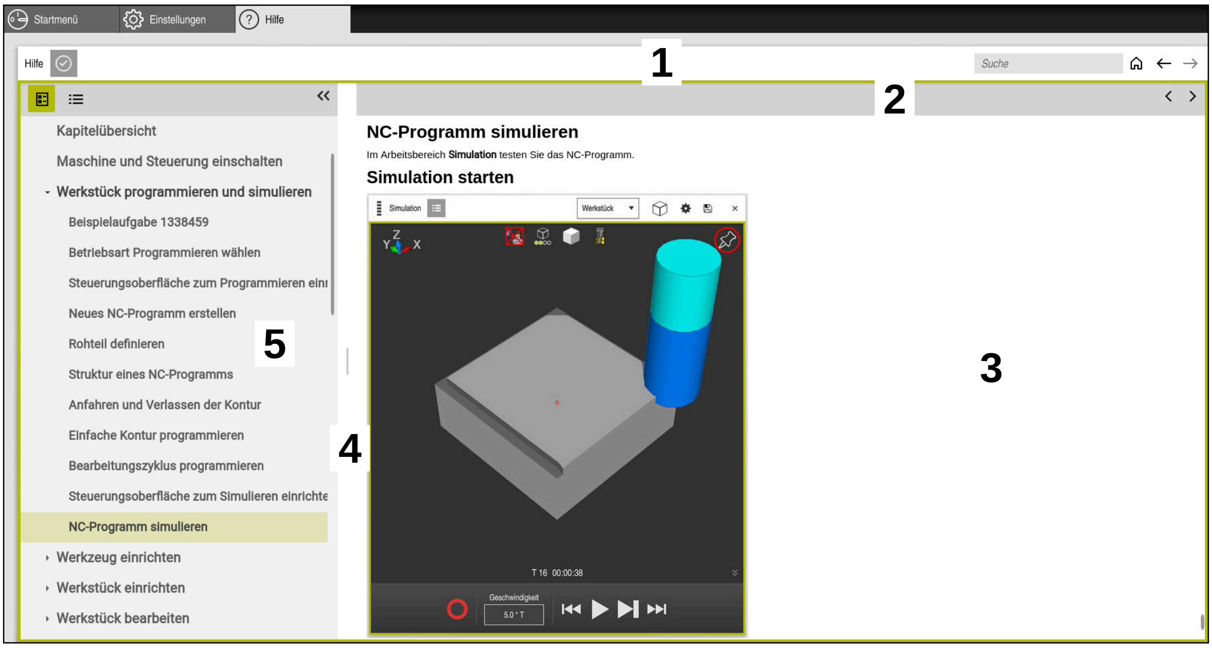
Further information: "Help application", Page 63

Further information: "Help workspace", Page 1378

Operation of **TNCguide** is identical in both cases.

Further information: "Symbols", Page 64

Help application



Help application with open **TNCguide**

The **Help** application includes the following areas:

1 Title bar in the **Help** application
  Further information: "Symbols in the Help application", Page 64
2 Title bar of the integrated **TNCguide** product help
  Further information: "Symbols in the integrated TNCguide product help ", Page 64
3 Content column of **TNCguide**
4 Separator between the columns of **TNCguide**
  Adjust the column width by means of the separator.
5 Navigation column of **TNCguide**

Symbols

Symbols in the Help application

Symbol	Function
	Show start page The start page displays all available documentation. Select the desired documentation, using a navigation tiles, e.g. **TNCguide**. If only one piece of documentation is available, the control opens the content directly. When a documentation is open, you can use the search function.
	Displaying tutorials
← →	Navigating between the last opened contents
	Displaying or hiding search results **Further information:** "Search in TNCguide", Page 65

Symbols in the integrated TNCguide product help

Symbol	Function
	Display documentation structure The structure consists of the content headings. The structure serves for main navigation within the documentation.
	Display documentation index The index consists of important keywords. The index serves as an alternative navigation within the documentation.
< >	Display previous or next page within the documentation
« »	Display or hide the navigation
	Copy NC examples to clipboard **Further information:** "Copying NC examples to clipboard", Page 65

1.5.1 Search in TNCguide

Using the search function, you can search for the entered search terms within the open documentation.

Use the search function as follows:

- Enter a character string

> The entry field is located in the title bar, to the left of the Home symbol that you use for navigating to the start page.
>
> The search starts automatically after you have entered e.g. a letter.
>
> If you wish to delete the entry, use the X symbol within the entry field.

- > The control opens the column containing the search results.
- > The control marks references also within open content pages.
- Select the reference
- > The control opens the selected content.
- > The control continues displaying the results of the last search.
- Select an alternative reference if necessary
- Enter a new character string if required

1.5.2 Copying NC examples to clipboard

Use the copy function to copy NC examples from the documentation to the NC editor.

To use the copy function:

- Navigate to the desired NC example
- **Notes on using NC programs** Expand
- **Notes on using NC programs** Read and pay attention to

  Further information: "Notes on using NC programs", Page 61

- Copy NC example to clipboard
- > The button switches colors while copying.
- > The clipboard contains the entire content of the copied NC example.
- Insert the NC example into the NC program
- Adapt the inserted contents according to the **Notes on using NC programs**
- Use the Simulation mode to test the NC program

  Further information: "Simulation Workspace", Page 1405

1.6 Contact to the editorial staff

Have you found any errors or would you like to suggest changes?

We continuously strive to improve our documentation for you. Please help us by sending your suggestions to the following e-mail address:

tnc-userdoc@heidenhain.de

2

About the Product

2.1 The TNC7

Every HEIDENHAIN control supports you with dialog-guided programming and finely detailed simulation. The TNC7 additionally offers you graphical or form-based programming to reach the desired result safe and sound.

Software options and optional hardware extensions can be used for flexibly increasing the range of functions and ease of use.

Such an extension provides e.g. the chance to perform turning and grinding in addition to milling and drilling processes.

Further information: "Technology-Specific NC Programming", Page 209

The ease of use increases e.g. when using touch probes, handwheels or a 3D mouse.

Further information: "Hardware", Page 82

Definitions

Abbreviation	Definition
TNC	**TNC** is derived from the acronym **CNC** (computerized numerical control). The **T** (tip or touch) represents the possibility to enter NC programs directly at the control or to program them graphically using gestures.
7	The product number indicates the control generation. The range of functions depends on the enabled software options.

2.1.1 Proper and intended operation

The information about the proper and intended operation supports you as user in the safe handling of a product such as a machine tool.

The control is a machine component but not a complete machine. This User's Manual describes the use of the control. Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.

> HEIDENHAIN sells controls designed for milling and turning machines as well as for machining centers with up to 24 axes. If you as a user face a different constellation, then contact the owner immediately.

HEIDENHAIN contributes additionally to enhancing your safety and that of your products, notably by taking into consideration the customer feedback. This results e.g. in function adaptations of the controls and safety precautions in the information products.

> Contribute actively to increasing the safety by reporting any missing or misleading information.
>
> **Further information:** "Contact to the editorial staff", Page 65

2.1.2 Intended place of operation

In accordance with DIN EN 50370-1 standard referring to electromagnetic compatibility (EMC), the control is approved for use in industrial environments.

Definitions

Guideline	Definition
DIN EN 50370-1:2006-02	This standard deals, among other things, with interference emissions and immunity to interference of machine tools.

2.2 Safety precautions

Comply with all safety precautions indicated in this document and in your machine manufacturer's documentation!

The following safety precautions refer exclusively to the control as an individual component but not to the specific complete product, i.e. the machine tool.

> Refer to your machine manual.
> Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.

The following overview contains exclusively the generally valid safety precautions. Pay attention to additional safety precautions that may vary with the configuration and are given in the following chapters.

> For ensuring maximum safety, all safety precautions are repeated at the relevant places within the chapters.

DANGER

Caution: hazard to the user!

Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!

- Devices should be connected or removed only by authorized service technicians
- Only switch on the machine via a connected handwheel or a secured connection

DANGER

Caution: hazard to the user!

Machines and machine components always pose mechanical hazards. Electric, magnetic, or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

- Read and follow the machine manual
- Read and follow the safety precautions and safety symbols
- Use the safety devices

DANGER

Caution: hazard to the user!

The **AUTOSTART** function automatically starts the machining operation. Open machines with unsecured work envelopes pose a huge danger for the machine operator.

- Use the **AUTOSTART** function exclusively on enclosed machines

WARNING

Caution: hazard to the user!

Manipulated data records or software can lead to an unexpected behavior of the machine. Malicious software (viruses, Trojans, malware, or worms) can cause changes to data records and software.

- Check any removable memory media for malicious software before using them
- Start the internal web browser only from within the sandbox

NOTICE

Danger of collision!

Failure to notice deviations between the actual axis positions and those expected by the control (saved at shutdown) can lead to undesirable and unexpected axis movements. There is risk of collision during the reference run of further axes and all subsequent movements!

- Check the axis positions
- Only confirm the pop-up window with **YES** if the axis positions match
- Despite confirmation, at first only move the axis carefully
- If there are discrepancies or you have any doubts, contact your machine tool builder

NOTICE

Caution: Danger to the tool and workpiece!

A power failure during the machining operation can cause uncontrolled "coasting" or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last saved axis values as the current position, which can deviate from the actual position. Thus, subsequent traverse movements do not correspond to the movements prior to the power failure. If the tool is still in effect during the traverse movements, then the tool and the workpiece can sustain damage through tension!

- Use a low feed rate
- Please keep in mind that the traverse range monitoring is not available for non-referenced axes

NOTICE

Danger of collision!

With NC software 81762x-16, the TNC7 does not support ISO programming. There is a risk of collision during execution due to the missing support.

- Use Klartext NC programs exclusively.

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision when referencing the axes.

- Pay attention to the information on the screen
- If necessary, move to a safe position before referencing the axes
- Watch out for possible collisions

NOTICE

Danger of collision!

The control uses the defined tool length from the tool table for compensating the tool length. Incorrect tool lengths will result in an incorrect tool length compensation. The control does not perform tool length compensation or a collision check for tools with a length of **0** and after **TOOL CALL 0**. Danger of collision during subsequent tool positioning movements!

- Always define the actual tool length of a tool (not just the difference)
- Use **TOOL CALL 0** only to empty the spindle

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

- Before activating a preset, check whether all columns contain values.

NOTICE

Danger of collision!

NC programs that were created on older controls can lead to unexpected axis movements or error messages on current control models. Danger of collision during machining!

- Check the NC program or program section using the graphic simulation
- Carefully test the NC program or program section in the **Program run, single block** operating mode
- Pay attention to the following known differences (the list below might not be complete!)

NOTICE

Caution: Data may be lost!

The **DELETE** function permanently deletes the file. The control does not perform an automatic backup of the file prior to deletion (e.g., there is no recycle bin). Files are thereby irreversibly deleted.

- Regularly back up important data to external drives

NOTICE

Caution: Data may be lost!

Never remove a connected USB device during data transfer—data can be damaged or deleted!

- Use the USB port for data transfer and backup only; do not use it for editing and executing NC programs
- Use the soft key to remove a USB device when data the transfer is complete

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss no matter what state the control was in!

- Always shut down the control
- Only operate the main switch after being prompted on the screen

NOTICE

Danger of collision!

If you select an NC block in program run using the **GOTO** function and then execute the NC program, the control ignores all previously programmed NC functions, e.g. transformations. This means that there is a risk of collision during subsequent traversing movements!

- Use **GOTO** only when programming and testing NC programs
- Only use **Block scan** when executing NC programs

2.3 Software

This User's Manual describes the functions for setting up the machine as well as for programming and and running your NC programs. These functions are available for a control featuring the full range of functions.

> The actual range of functions depends, among other things, on the enabled software options.
> **Further information:** "Software options", Page 74

The table shows the NC software numbers described in this User's Manual.

> HEIDENHAIN has simplified the version schema, starting with NC software version 16:
> - The publication period determines the version number.
> - All control models of a publication period have the same version number.
> - The version number of the programming stations corresponds to the version number of the NC software.

NC software number	Product
817620-16	TNC7
817621-16	TNC7 E
817625-16	TNC7 programming station

> Refer to your machine manual.
> This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine.
> Check, on the basis of the machine tool manual, whether the machine manufacturer has adapted the functions of the control.

Definition

Abbreviation	Definition
E	The suffix E indicates the export version of the control. In this version, the software option 9 Advanced Function Set 2 is restricted to 4-axis interpolation.

2.3.1 Software options

Software options define the range of functions of the control. The optional functions are either machine- or application-specific. The software options give you the possibility of adapting the control to your individual needs.

You can check which software options are enabled on your machine.

Further information: "Viewing of software options", Page 1918

Overview and definitions

The **TNC7** features various software options, each of which can be enabled separately and even subsequently by the machine manufacturer. The following overview includes only those software options that are relevant for you as user.

> The option numbers indicated in the User's Manual show you that a function is not included in the standard range of available functions.
> The Technical Manual informs about additional software options that are relevant for the machine manufacturer.

> Keep in mind that particular software options also require hardware extensions.
> **Further information:** "Hardware", Page 82

Software option	Definition and application
Additional Axis (options 0 to 7)	**Additional control loop** A control loop is required for each axis or spindle moved to a programmed nominal value by the control. Additional control loops are required e.g. for detachable and motor-driven tilting tables.
Advanced Function Set 1 (option 8)	**Advanced functions (set 1)** On machines with rotary axes this software option enables the machining of multiple workpiece sides in a single setup. The software option includes the following functions: ■ Tilting the working plane, e.g. with **PLANE SPATIAL** **Further information:** "PLANE SPATIAL", Page 984 ■ Programming of contours on the unrolled surface of a cylinder, e.g. by using Cycle **27 CYLINDER SURFACE** **Further information:** "Cycle 27 CYLINDER SURFACE (option 8)", Page 1178 ■ Programming the rotary axis feed rate in mm/min with **M116** **Further information:** "Interpreting the feed rate for rotary axes as mm/min with M116 (option 8)", Page 1238 ■ 3-axis circular interpolation with a tilted working plane The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.
Advanced Function Set 2 (option 9)	**Advanced functions (set 2)** On machines with rotary axes this software option enables the simultaneous 5-axis machining of workpieces. The software option includes the following functions: ■ **TCPM** (tool center point management): Automatic tracking of linear axes during the rotary axis positioning **Further information:** "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027 ■ Running of NC programs with vectors, including optional 3D tool compensation **Further information:** "3D tool compensation (option 9)", Page 1049 ■ Manual moving of axes in the active tool coordinate system **T-CS** ■ Linear interpolation in more than 4 axes (max. 4 axes in case of an export version) The advanced functions (set 2) can be used to produce free-form surfaces.

Software option	Definition and application
HEIDENHAIN DNC (option 18)	**HEIDENHAIN DNC** This software option enables external Windows applications to access data of the control via the TCP/IP protocol. Potential fields of application are e.g.: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data HEIDENHAIN DNC is required in conjunction with external Windows applications.
Dynamic Collision Monitoring (option 40)	**Dynamic Collision Monitoring (DCM)** The machine manufacturer can use this software option to define machine components as collision objects. The control monitors the defined collision objects during all machine movements. The software option includes the following functions: ■ Automatic interruption of the program run whenever a collision is imminent. ■ Warnings in case of manual axis movements ■ Collision monitoring in Test Run mode With DCM you can prevent collisions and thus avoid additional costs resulting from material damage or a machine downtime. **Further information:** "Dynamic Collision Monitoring (DCM, option 40)", Page 1084
CAD Import (option 42)	**CAD Import** This software option is used to select positions and contours from CAD files and to transfer them into an NC program. With the CAD Import option you reduce the programming effort and prevent typical errors such as the incorrect entry of values. In addition, CAD Import contributes to paperless manufacturing. "Applying contours and positions to NC programs with CAD Import (option 42)"
Global PGM Settings (option 44)	**Global Program Settings GPS** This software option can be used for superimposed coordinate transformations and handwheel movements during program run without changing the NC program. With GPS you can adapt externally created NC programs to the machine and increase flexibility during program run. **Further information:** "Globale Programmeinstellungen GPS", Page
Adaptive Feed Control (option 45)	**Adaptive Feed Control AFC** This software option enables an automatic feed control that depends on the current spindle load. The control increases the feed rate as the load decreases and reduces the feed rate as the load increases. With AFC you can shorten the machining time without adapting the NC program, while preventing machine damage from overload at the same time. **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 1114

Software option	Definition and application
KinematicsOpt (option 48)	**KinematicsOpt** This software option uses automatic probing processes to check and optimize the active kinematics. With KinematicsOpt the control can correct position errors on rotary axes and thus increase the accuracy of machining operations in the tilted working plane and of simultaneous machining operations. In part, the control can compensate temperature-induced deviations through repeated measurements and corrections. **Further information:** "Touch Probe Cycles: Automatic Kinematics Measurement", Page 1695
Turning (option 50)	**Mill-turning** This software option offers a comprehensive milling-specific function package for milling machines with rotary tables. The software option includes the following functions: ▪ Turning-specific tools ▪ Turning-specific cycles and contour elements such as undercuts ▪ Automatic tool radius compensation Mill-turning enables mill-turning machining operations on only one machine, thus reducing e.g. the setup work effort considerably. **Further information:** "Turning (option 50)", Page 212
KinematicsComp (option 52)	**KinematicsComp** This software option uses automatic probing processes to check and optimize the active kinematics. With KinematicsComp the control can correct position and component errors in three dimensions. This means it can compensate the errors of rotary and linear axes in three dimensions. Compared to KinematicsOpt (option 48), the corrections are even far more comprehensive. **Further information:** "Cycle 453 KINEMATICS GRID (option 48), (option 52)", Page 1727
OPC UA NC Server 1 to **6** (options 56 to 61)	**OPC UA NC Server** The software options offer the OPC UA standardized interface for the external access to data and functions of the control. Potential fields of application are e.g.: ▪ Connection to higher-level ERP or MES systems ▪ Capture of machine and operating data Each software option enables one client connection each. Several parallel connections require the use of multiple OPC UA NC servers. **Further information:** "OPC UA NC Server (options 56 to 61)", Page 1931
4 Additional Axes (option 77)	**4 additional control loops** see "**Additional Axis** (options 0 to 7)"
8 Additional Axes (option 78)	**8 additional control loops** see "**Additional Axis** (options 0 to 7)"

Software option	Definition and application
3D-ToolComp (option 92)	**3D-ToolComp** only in connection with Advanced Function Set 2 (option 9) With this software option, shape deviations on ball cutters and workpiece probes can be automatically compensated using a correction value table. 3D-ToolComp enables increasing the workpiece accuracy in conjunction with free-form surfaces, for example. **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1062
Extended Tool Management (option 93)	**Extended tool management** This software option extends tool management by the two tables **Tooling list** and **T usage order**. The tables show the following contents: ■ The **Tooling list** shows the tool requirements of the NC program or pallet to be run **Further information:** "Tooling list (option 93)", Page 1849 ■ The **T usage order** shows the tool order of the NC program or pallet to be run **Further information:** "T usage order (option 93)", Page 1847 The extended tool management enables you to detect the tool requirements in time and thus prevent interruptions during program run.
Advanced Spindle Interpolation (option 96)	**Interpolating Spindle** This software option enables interpolation turning, as the control couples the tool spindle with the linear axes. The software option includes the following cycles: ■ Cycle **291 COUPLG.TURNG.INTERP.** for simple turning machining operations without contour subprograms **Further information:** "Cycle 291 COUPLG.TURNG.INTERP. (option 96)", Page 621 ■ Cycle **292 CONTOUR.TURNG.INTRP.** for finishing rotationally symmetrical contours **Further information:** "Cycle 292 CONTOUR.TURNG.INTRP. (option 96)", Page 628 The interpolating spindle enables you to execute a turning operation also on machines without rotary table.
Spindle Synchronism (option 131)	**Spindle synchronism** This software option synchronizes two or more spindles and thus enables e.g. the manufacture gears by hobbing. The software option includes the following functions: ■ Spindle synchronism for special machining operation, e.g. polygonal turning ■ Cycle **880 GEAR HOBBING** only in conjunction with mill-turning (option 50) **Further information:** "Cycle 880 GEAR HOBBING (option 131)", Page 898
Remote Desktop Manager (option 133)	**Remote Desktop Manager** This software option is used to display and operate externally linked computer units. With Remote Desktop Manager you reduce the distances covered between several workplaces and as a result increase the efficiency. **Further information:** "Remote Desktop Manager window (option 133)", Page 1944

Software option	Definition and application
Dynamic Collision Monitoring v2 (option 140)	**Dynamic Collision Monitoring (DCM) version 2** This software option includes the functions of software option 40 (Dynamic Collision Monitoring, DCM). In addition, this software option can be used for the collision monitoring of workpiece fixtures. **Further information:** "Integrating the fixtures into collision monitoring (option 140)", Page 1094
Cross Talk Compensation (option 141)	**Compensation of axis couplings CTC** Using this software option, the machine manufacturer can e.g. compensate acceleration-induced deviations at the tool and thus increase accuracy and dynamic performance.
Position Adaptive Control (option 142)	**Position adaptive control PAC** Using this software option, the machine manufacturer can e.g. compensate position-induced deviations at the tool and thus increase accuracy and dynamic performance.
Load Adaptive Control (option 143)	**Load adaptive control LAC** Using this software option, the machine manufacturer can e.g. compensate load-induced deviations at the tool and thus increase accuracy and dynamic performance.
Motion Adaptive Control (option 144)	**Motion adaptive control MAC** Using this software option, the machine manufacturer can e.g. change speed-dependent machine settings and thus increase the dynamic performance.
Active Chatter Control (option 145)	**Active chatter control ACC** With this software option the chatter tendency of a machine used for heavy machining can be reduced. The control can use ACC to improve the surface quality of the workpiece, increase the tool life and reduce the machine load. Depending on the type of machine, the metal-removal rate can be increased by more than 25 %. **Further information:** "Active Chatter Control (ACC, option 145)", Page 1121
Machine Vibration Control (option 146)	**Vibration damping for machines MVC** Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ AVD **Active Vibration Damping** ■ FSC **Frequency Shaping Control**
CAD Model Optimizer (option 152)	**Optimization of CAD models** This software option can be used, for example, to repair faulty files of fixtures and tool holders or to position STL files generated from the simulation for a different machining operation. **Further information:** "Generating STL files with 3D mesh (option 152)", Page 1372

Software option	Definition and application
Batch Process Manager (option 154)	**Batch Process Manager BPM** This software option makes it easy to plan and execute multiple production jobs. If pallet management and extended tool management (option 93) are extended or combined, BPM offers the following additional data, for example: ■ Machining time ■ Availability of necessary tools ■ Manual interventions to be made ■ Program test results of assigned NC programs **Further information:** "Job list workspace", Page 1764
Component Monitoring (option 155)	**Component monitoring** This software option enables the automatic monitoring of machine components configured by the machine manufacturer. Component monitoring assists the control in preventing machine damage due to overload by way of hazard warnings and error messages.
Grinding (option 156)	**Jig grinding** This software option offers a comprehensive grinding-specific function package for milling machines. The software option includes the following functions: ■ Grinding-specific tools including dressing tools ■ Cycles for reciprocating stroke and dressing Jig-turning enables complete machining operations on just one machine, thus reducing e.g. the setup work effort considerably. **Further information:** "Grinding operations (option 156)", Page 224
Gear Cutting (option 157)	**Gear manufacturing** This software option enables the manufacturing of cylindrical gears or helical gears of any angle. The software option includes the following cycles: ■ Cycle **285 DEFINE GEAR** to define the gear geometry **Further information:** "Cycle 285 DEFINE GEAR (option 157)", Page 909 ■ Cycle **286 GEAR HOBBING** **Further information:** "Cycle 286 GEAR HOBBING (option 157)", Page 911 ■ Cycle **287 GEAR SKIVING** **Further information:** "Cycle 287 GEAR SKIVING option 157", Page 919 Gear manufacturing expands the scope of functionality of milling machines with rotary tables even without mill-turning (option 50).

Software option	Definition and application
Turning v2 (option 158)	**Mill-turning version 2** This software option includes all functions of Mill-Turning (software option 50). In addition, this software option offers the following advanced turning functions: ■ Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** **Further information:** "Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING (option158) ", Page 816 ■ Cycle **883 TURNING SIMULTANEOUS FINISHING** **Further information:** "Cycle 883 TURNING SIMULTANEOUS FINISHING (option 158)", Page 821 The advanced turning functions not only enable you to manufacture undercut workpieces but also to use a larger area of the indexable insert during the machining operation.
Optimized Contour Milling (option 167)	**Optimized contour machining (OCM)** This software option enables the trochoidal milling of closed or open pockets and islands of any shape. During trochoidal milling, the full cutting edge is used under constant cutting conditions. The software option includes the following cycles: ■ Cycle **271 OCM CONTOUR DATA** ■ Cycle **272 OCM ROUGHING** ■ Cycle **273 OCM FINISHING FLOOR** and cycle **274 OCM FINISHING SIDE** ■ Cycle **277 OCM CHAMFERING** ■ In addition, the control offers **OCM FIGURES** for frequently needed contours With OCM you can shorten the machining time while reducing tool wear at the same time. **Further information:** "OCM cycles", Page 594
Process Monitoring (option 168)	**Process monitoring** Reference-based monitoring of the machining process The control uses this software option to monitor defined machining sections during program run. The control compares changes in conjunction with the tool spindle or the tool with the values of a reference machining operation. **Further information:** "Arbeitsbereich Prozessüberwachung (Option #168)", Page

2.3.2 Feature Content Level

New functions or functional enhancements of the control software may be protected by software options or by means of the Feature Content Levels.

On purchasing a new control you will receive the highest level of the **FCL** possible with the installed software. A subsequent software update e.g. within the frame of a service requirement, does not automatically increase the **FCL** version.

> As yet, no functions are protected by Feature Content Level. If functions are protected in future, the User's Manual will indicate the marking **FCL n**. The **n** shows the required number of the **FCL** version.

2.3.3 Information on licensing and use

Open-source software

The control software contains open-source software whose use is subject to explicit licensing terms. These special terms of use have priority.

To get to the licensing terms on the control:

- Select the **Home** operating mode
- Select the **Settings** application
- Select the **Operating system** tab
- Double-tap or double-click **About HeROS**
- > The control opens the **HEROS Licence Viewer** window.

OPC UA

The control software contains binary libraries. For these libraries, additionally and preferentially the terms of use agreed between HEIDENHAIN and Softing Industrial Automation GmbH shall apply.

OPC UA NC Server (options 56 to 61) and HEIDENHAIN DNC (option 18) can be used to influence the behavior of the control. Before using these interfaces for productive purposes, system tests must be performed to exclude the occurrence of any malfunctions or performance failures of the control. The manufacturer of the software product that uses these communication interfaces is responsible for performing these tests.

Further information: "OPC UA NC Server (options 56 to 61)", Page 1931

2.4 Hardware

This User's Manual describes functions for setting up and operating the machine. These functions primarily depend on the installed software.

Further information: "Software", Page 74

The actual range of functions also depends on hardware enhancements and the enabled software options.

2.4.1 Monitor

BF 360

The TNC7 is delivered with a 24-inch touchscreen.

The control is operated by means of touchscreen gestures and with the operating elements of the keyboard unit.

Further information: "Common gestures for the touchscreen", Page 96

Further information: "Operating elements of the keyboard unit", Page 96

Operation and cleaning

> **Avoiding electrostatic discharge when operating touchscreens**
> Touchscreens are based on a capacitive working principle, i.e. they are sensitive to electrostatic charges generated by the operators.
> Users can discharge static electricity from their body by touching grounded metal objects. This problem can be avoided by wearing ESD clothing.

Capacitative sensors detect a contact as soon as a person’s finger touches the touchscreen. Touchscreens can even be operated with dirty hands, as long as the touch sensors are able to detect the skin resistance. While small amounts of liquid will not cause a fault, larger quantities of liquid will cause erroneous input.

> Use work gloves to prevent the device from becoming dirty. The rubber material of special touchscreen work gloves contains metal ions that transfer the skin resistance to the display.

In order to maintain the functionality of the touchscreen, use the following cleaners only:
- Glass cleaner
- Foaming screen cleaners
- Mild detergents

> Do not apply the cleaner directly to screen, but slightly dampen a suitable cleaning cloth with it.

Switch off the control before cleaning the touchscreen. As an alternative, you can use the touchscreen cleaning mode.
Further information: "Settings Application", Page 1911

> Never use the following cleaners or cleaning aids, in order to avoid damage to the touchscreen:
> - Aggressive solvents
> - Abrasives
> - Compressed air
> - Steam cleaners

2.4.2 Keyboard unit

TE 360 with standard potentiometer layout

TE 360 with alternative potentiometer layout

TE 361

The TNC7 is delivered with various keyboard units.

The control is operated by means of touchscreen gestures and with the operating elements of the keyboard unit.

Further information: "Common gestures for the touchscreen", Page 96

Further information: "Operating elements of the keyboard unit", Page 96

Refer to your machine manual.

Some machine tool builders do not use the standard HEIDENHAIN operating panel.

External keys, e.g. **NC START** or **NC STOP**, are described in your machine manual.

Cleaning

> Use operating gloves to prevent the device from becoming dirty.

In order to maintain the functionality of the keyboard, use only cleaners stated to contain anionic or nonionic surfactants.

> Do not apply the cleaner directly to the keyboard unit. Slightly dampen a suitable cleaning cloth with the cleaner.

Switch the control off before cleaning the keyboard unit.

> Never use the following cleaners or cleaning aids, in order to avoid damage to the keyboard unit:
> - Aggressive solvents
> - Abrasives
> - Compressed air
> - Steam blasters

> The trackball does not require periodic maintenance. Cleaning is required only if the trackball stops functioning.

If a trackball is embedded in the keyboard, clean the trackball as follows:

- ▶ Switch off the control
- ▶ Turn the pull-off ring by 100° in counterclockwise direction
- > Turning the removable pull-off ring moves it upwards out of the keyboard unit.
- ▶ Remove the pull-off ring
- ▶ Take out the ball
- ▶ Carefully remove sand, chips, or dust from the shell area

> Scratches in the shell area may impair the functionality or prevent proper functioning.

- ▶ Apply a small amount of an isopropyl alcohol cleaner to a lint-free and clean cloth

> Please observe the information for the cleaner.

- ▶ Carefully wipe the shell area clean with the cloth until all smears or stains have been removed

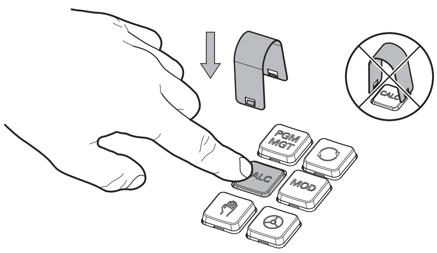
Exchanging keycaps

If you need replacements for the keycaps of the keyboard unit, contact HEIDENHAIN or the machine manufacturer.

Further information: "Keycaps for keyboard units and machine operating panels", Page 2087

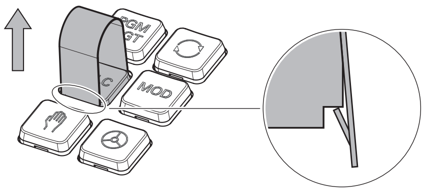
The IP54 protection rating cannot be guaranteed if the keyboard is missing any keys.

To exchange the keycaps:

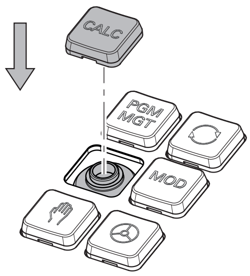


- Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage

Pressing the key will make it easier to apply the keycap puller.



- Pull off the keycap



- Place the keycap onto the seal and push it down

The seal must not be damaged; otherwise the IP54 protection rating cannot be guaranteed.

- Verify proper seating and correct functioning

2.4.3 Hardware enhancements

The hardware enhancements give you the possibility of adapting the machine tool to your individual needs.

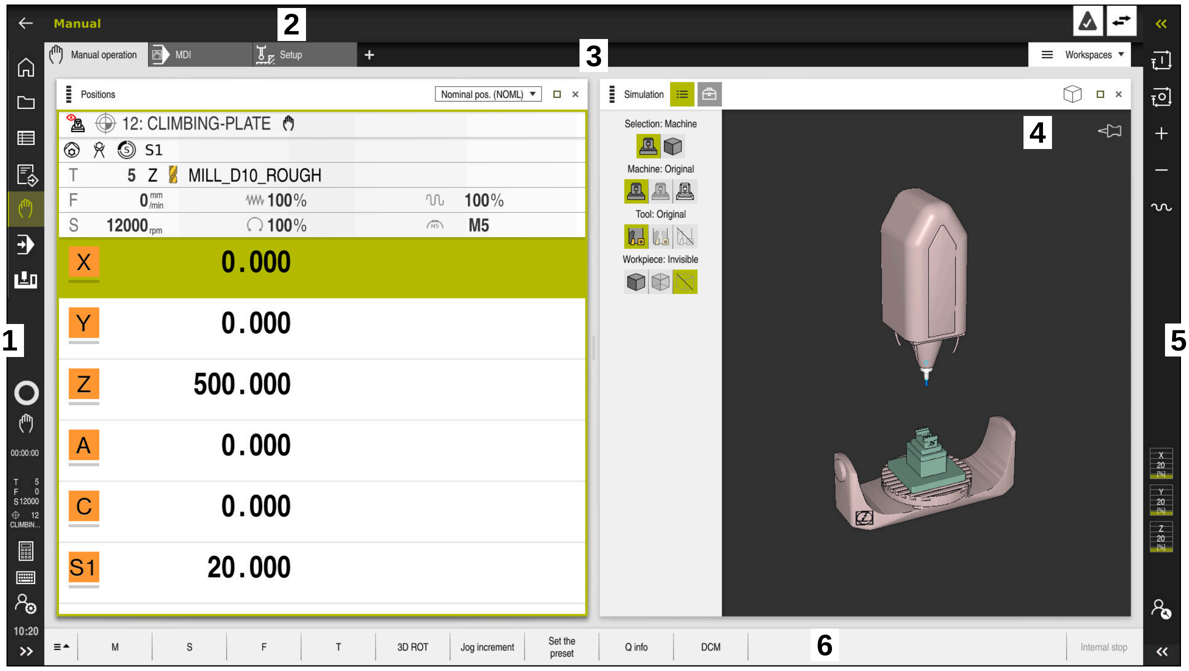
The **TNC7** features various hardware enhancements, each of which can be added separately and even subsequently by the machine manufacturer. The following overview includes only those enhancements that are relevant for you.

> Keep in mind that particular hardware enhancements require additional software options.
>
> **Further information:** "Software options", Page 74

Hardware enhance-ments	Definition and application
Electronic handwheels	You use this enhancement for exact manual positioning of machine axes. The wireless portable variants improve ergonomics and increase versatility. The handwheels have the following differing features: ■ Portable or installed in the machine operating panel ■ With or without display ■ With or without functional safety Electronic handwheels, for example, greatly simplify workpiece setup. **Further information:** "Electronic Handwheel", Page 1881
Workpiece touch probes	The control uses this enhancement for automatic and precise detection of workpiece positions and misalignments. The workpiece touch probes have the following differing features: ■ With radio or infrared transmission ■ With or without cable Workpiece touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions during program run. **Further information:** "Touch Probe Functions in the Manual Operating Mode", Page 1425
Tool touch probes	The control uses this enhancement for automatic and precise calibration of tools directly in the machine. Tool touch probes have the following differing features: ■ Contact-free or tactile measurement ■ With radio or infrared transmission ■ With or without cable Tool touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions and breakage control during program run. "Touch Probe Cycles: Automatic Tool Measurement"
Vision systems	Use this enhancement to inspect the tools used. With the VT 121 vision system, you can visually inspect the cutting edges during program run without removing the tool. The vision systems help to avoid damage during program run, thus preventing unnecessary costs.

<table>
<tr><th>Hardware enhancements</th><th>Definition and application</th></tr>
<tr><td>Additional operating stations</td><td>This enhancement adds a second screen, to facilitate operation of the control.
The additional ITC (industrial thin client) operating stations are differentiated by their intended use:
■ The ITC 755 is a compact, additional operating station that mirrors the control's main screen, making it possible to operate the control.
■ The ITC 750 and ITC 860 are additional screens that increase the display area of the main screen, so that you can view multiple applications at the same time.
By adding a keyboard unit, the ITC 750 and ITC 860 can be used as full-fledged additional operating stations.
The additional operating stations increase operator comfort, especially on large machining centers.</td></tr>
<tr><td>Industrial PC</td><td>You use this enhancement to install and run Windows-based applications.
With Remote Desktop Manager (option 133), you can display the applications on the control’s screen.
Further information: "Remote Desktop Manager window (option 133)", Page 1944
The industrial PC is a secure and powerful alternative to external PCs.</td></tr>
</table>

2.5 Areas of the control's user interface



The control's user interface in the **Manual operation** application

The control's user interface shows the following areas:

1 TNC bar
- Back
  Use this function to go backwards in the application history since booting the control.
- Operating modes
  Further information: "Overview of operating modes", Page 90
- Status overview
  Further information: "Status overview on the control bar", Page 147
- Calculator
  Further information: "Calculator", Page 1396
- Screen keyboard
  Further information: "Virtual keyboard of the control bar", Page 1380
- Settings
  In the settings you can choose among several pre-defined views for the control's user interface.
- Date and time

2 Information bar
- Active operating mode
- Message menu
  Further information: "Message menu on the information bar", Page 1400
- Symbols

3 Application bar

- Tabs of opened applications
- Selection menu for workspaces

  With the selection menu you define which workspaces are open in the active application.

4 Workspaces

Further information: "Workspaces", Page 92

5 Machine manufacturer bar

The machine manufacturer configures the machine manufacturer bar.

6 Function bar

- Selection menu for buttons

  With the selection menu you define which buttons the control displays in the function bar.
- Button

  With the buttons you activate individual functions of the control.

2.6 Overview of operating modes

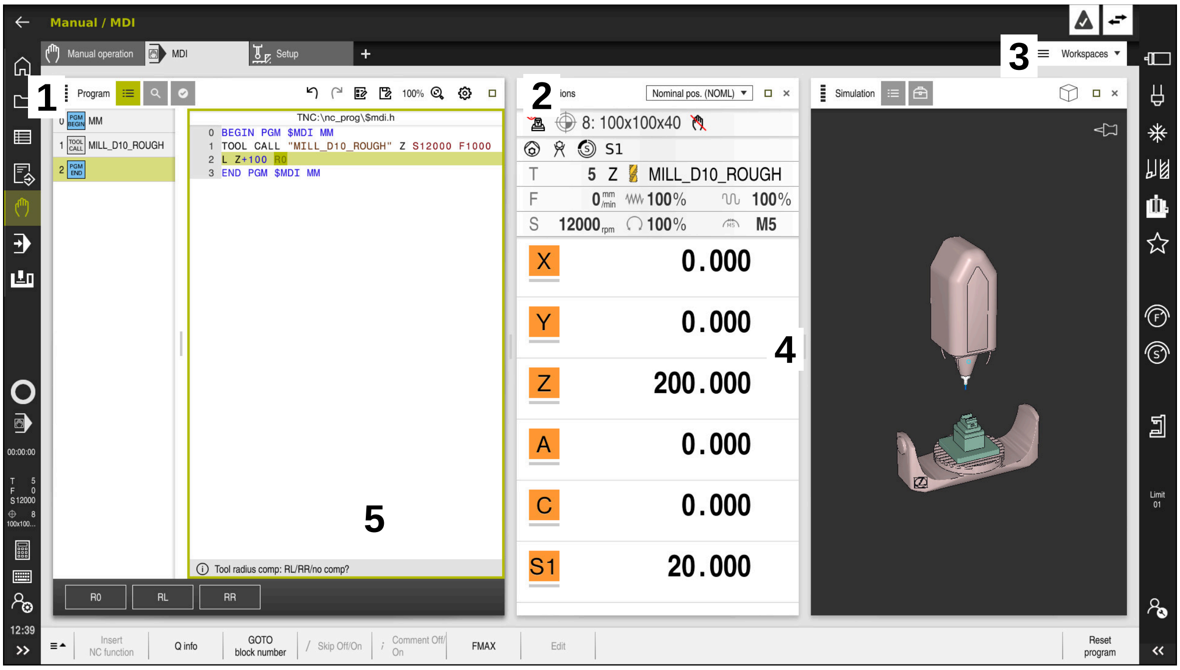
The control provides the following operating modes:

Symbols	Operating modes	Further information
	The **Home** operating mode contains the following applications: ■ **Start/Login** application During the startup process, the control is in the **Start/Login** application.	
	■ **Settings** application	Page 1911
	■ **Help** application	
	■ Applications for machine parameters	Page 1958
	In the **Files** operating mode the control displays drives, folders and files. You can, for example, create or delete folders or files and can also connect drives.	Page 1066
	In the **Tables** operating mode you can open various tables and edit them as necessary.	Page 1802
	In the **Editor** operating mode you can do the following: ■ Create, edit and simulate NC programs ■ Create and edit contours ■ Create and edit pallet tables	
	The **Manual** operating mode contains the following applications:	
	■ **Manual operation** application	Page 180
	■ **MDI** application	Page 1759
	■ **Setup** application	Page 1425
	■ **Move to ref. point** application	Page 174

Symbols	Operating modes	Further information
	In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either one block at a time or in full sequence. You also execute pallet tables in this operating mode.	Page 1778
	In the **Retract** application you can move the tool away from the workpiece, for example after a power failure.	Page 1796
	If the machine manufacturer has defined an embedded workspace, then you can open full-screen mode with this operating mode. The machine manufacturer defines the name of the operating mode. Refer to your machine manual.	Page 1899
	In the **Machine** operating mode the machine manufacturer defines his own functions, such as diagnostic functions for spindle and axes, or other applications. Refer to your machine manual.	

2.7 Workspaces

2.7.1 Operating elements within the workspaces



The control in the **MDI** application with three open workspaces

The control displays the following operating elements:

1 Gripper
   Use the gripper in the title bar to change positions of the workspaces. You can also align two workspaces vertically above each other.
2 Title bar
   In the title bar the control shows the title of the workspace, and different symbols or settings, depending on the workspace.
3 Selection menu for workspaces
   Use the selection menu for workspaces in the application bar to open individual workspaces. The available workspaces depend on the active application.
4 Separator
   You use the separator between two workspaces to change the scaling of the workspaces.
5 Action bar
   In the action bar the control shows selection possibilities for the current dialog; for example, an NC function.

2.7.2 Symbols within the workspaces

If more than one workspace is open, the title bar contains the following symbols:

Icon	Function
	Maximize workspace
	Reduce workspace
	Close workspace

If you maximize a workspace, the control shows the workspace over the application's entire area. If you reduce the workspace, then all other workspaces return to their previous position.

2.7.3 Overview of workspaces

The control offers the following workspaces:

Workspace	Further information
Probing function In the **Probing function** workspace you set presets on the workpiece and determine and compensate for workpiece misalignment and rotations. You can also calibrate the touch probe, measure tools, and set up fixtures.	Page 1425
Job list In the **Job list** workspace, you edit and execute pallet tables.	Page 1764
Open File The **Open File** workspace allows e. g. selecting and creating files.	Page 1075
Form for tables In the **Form** workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.	Page 1807
Form for pallets In the **Form** workspace, the control shows the contents of the pallet table for the selected row.	Page 1771
Retract In the **Retract** workspace you disengage the tool after a power interruption.	Page 1796
GS (option 44) In the **GS** workspace you define selected transformations and settings without modifying the NC program.	Page 1132
Desktop menu In the **Desktop menu** workspace the control displays selected control and HEROS functions.	Page 104
Help In the **Help** workspace the control displays a help graphic for the current syntax element of an NC function or the **TNCguide** integrated product help.	Page 1378

Workspace	Further information
Contour In the **Contour** workspace you use lines and arcs to draw a 2D sketch and then generate a Klartext contour from it. You can also import program sections with contours from an NC program to the **Contour** workspace for graphical editing.	Page 1337
List In the **List** workspace the control shows the machine parameter structure; you might be able to edit some of the parameters.	Page 1959
Positions In the **Positions** workspace the control displays information about the status of various functions of the control and about current axis positions.	Page 141
Program The control displays the NC program in the **Program** workspace.	Page 194
RDP (option 133) If the machine manufacturer has defined an embedded workspace, you can see and operate the screen of an external computer on the control. The machine manufacturer can change the name of the workspace. Refer to your machine manual.	Page 1899
Quick selection In the **Quick selection** workspace you open an existing table or create a file, e. g. an NC program.	Page 1076
Simulation In the **Simulation** workspace the control shows the simulated or actual movements, depending on the operating mode.	Page 1405
Simulation status In the **Simulation status** workspace the control shows data based on the simulation of the NC program.	Page 165
Start/Login In the **Start/Login** workspace the control shows the steps that are performed while booting.	Page 108
Status In the **Status** workspace the control shows the status and values of individual functions.	Page 149
Table In the **Table** workspace, the control shows the contents of a table. The control displays a column with filters and a search function on the left side of some tables.	Page 1803
Table for machine parameters In the **Table** workspace the control shows the machine parameters; you might be able to edit some of them.	Page 1959
Keyboard In the **Keyboard** workspace you can enter NC functions, letters and numbers, and also navigate.	Page 1380

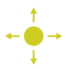
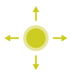
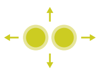
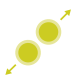
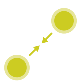
Workspace	Further information
Overview In the **Overview** workspace the control displays the information on the status of individual functional safety (FS) aspects.	Page 1906
Monitoring In the **Process Monitoring** workspace the control visualizes the machining process during program run. You can activate various monitoring tasks that are relevant to the process. If necessary, you can adapt the monitoring tasks.	Page 1153

2.8 Operating elements

2.8.1 Common gestures for the touchscreen

The screen of the control is multi-touch capable. That means the control can distinguish various gestures, even with two or more fingers at once.

You can use the following gestures:

Symbol	Gesture	Meaning
	Tap	A brief touch by a finger on the screen
	Double tap	Two brief touches on the screen
	Long press	Continuous contact of finger tip on the screen If you do not stop holding, the control will automatically cancel the holding gesture after approximately ten seconds. Permanent actuation is thus not possible.
	Swipe	Flowing motion over the screen
	Drag	A combination of long-press and then swipe, moving a finger over the screen when the starting point is clearly defined
	Two-finger drag	A combination of long-press and then swipe, moving two fingers in parallel over the screen when the starting point is clearly defined
	Spread	Two fingers long-press and move away from each other
	Pinch	Two fingers move toward each other

2.8.2 Operating elements of the keyboard unit

Application

You operate the **TNC7** primarily through the touchscreen, meaning with gestures.

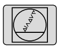
Further information: "Common gestures for the touchscreen", Page 96

In addition, the control's keyboard unit offers keys and other elements for alternative operating sequences.

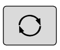
Description of function

The tables below describe the keyboard unit's operating elements.

Keycaps for alphabetic keyboard

Key	Function
A B C	Enter texts (e.g., file names)
SHIFT + Q	**Uppercase Q** If an NC program is open, in the **Editor** operating mode for entering a Q parameter formula; in the **Manual** operating mode for opening the **Q parameter list** window **Further information:** "Q parameter list window", Page 1271
ESC	Close windows and context menus
PRT SC	Create screenshot
	**Left DIADUR key** Open the **HEROS menu**
	In **Klartext programming** open a context menu

Keycaps for operating aids

Key	Function
PGM MGT	Open the **Open File** workspace in the **Editor** and **Program Run** operating modes **Further information:** "Open File workspace", Page 1075
	Select the button at the extreme right margin
ERR	Open and close the message menu **Further information:** "Message menu on the information bar", Page 1400
CALC	Open and close the calculator **Further information:** "Calculator", Page 1396
MOD	Open the **Settings** application **Further information:** "Settings Application", Page 1911
HELP	Open the online help **Further information:** "User's Manual as integrated product help: TNCguide", Page 62

Operating modes

i	On the TNC7 the operating modes of the control are allocated differently than on the TNC 640. For reasons of compatibility and to facilitate ease of operation, the keys on the keyboard unit remain the same. Keep in mind that particular keys no longer activate a change of operating modes but e.g. Instead activate a switch.

Key	Function
	Opening the **Manual operation** application in the **Manual** operating mode **Further information:** "Manual operation application", Page 180
	Activating and deactivating the electronic handwheel in the **Manual** operating mode **Further information:** "Electronic Handwheel", Page 1881
	Opening the **Tool Management** tab in the **Tables** operating mode **Further information:** "Tool management ", Page 270
	Opening the **MDI** application in the **Manual** operating mode **Further information:** "Application MDI", Page 1759
	Opening the **Program Run** operating mode in **Single Block** mode **Further information:** "Program Run operating mode", Page 1778
	Opening the **Program Run** operating mode **Further information:** "Program Run operating mode", Page 1778
	Opening the **Editor** operating mode **Further information:** "Editor operating mode", Page 192
	While the NC program is running, opening of the **Simulation** workspace in the **Editor** operating mode **Further information:** "Simulation Workspace", Page 1405

Keycaps for NC dialog

> The following functions are valid for the **Editor** operating mode and the **MDI** application.

Key	Function
APPR DEP	In the **Insert NC function** window, open the **Path contour** folder in order to select an approach or departure function **Further information:** "Approaching and departing from a contour", Page 317
FK	Open the **Contour** workspace (e.g., to draw a milling contour) Only in the **Editor** operating mode **Further information:** "Graphical Programming", Page 1337
CHF	Program a chamfer **Further information:** "Chamfer CHF", Page 297
L	Program a straight line segment **Further information:** "Straight line L", Page 297
CR	Program a circular arc with radius entry **Further information:** "Circular path CR", Page 302
RND	Program a rounding arc **Further information:** "Rounding RND", Page 299
CT	Program a circular arc with tangential connection to the preceding contour element **Further information:** "Circular path CT", Page 303
CC	Program a circle center or pole **Further information:** "Circle center point CC", Page 300
C	Program a circular arc with reference to the circle center **Further information:** "Circular path C ", Page 300
TOUCH PROBE	In the **Insert NC function** window, open the **Setup** folder in order to select a touch probe cycle **Further information:** "Programmable Touch Probe Cycles", Page 1449
CYCL DEF	In the **Insert NC function** window, open the **Cycles** folder in order to select a cycle **Further information:** "Defining cycles", Page 423
CYCL CALL	In the **Insert NC function** window, open the **Cycle call** folder in order to select a machining cycle **Further information:** "Calling cycles", Page 425
LBL SET	Program a jump label **Further information:** "Defining a label with LBL SET", Page 332
LBL CALL	Program a subprogram or a program section repeat **Further information:** "Calling a label with CALL LBL", Page 333

Key	Function
STOP	Program an intentional stop **Further information:** "Programming the STOP function", Page 1224
TOOL DEF	Pre-select a tool in the NC program **Further information:** "Tool pre-selection by TOOL DEF", Page 283
TOOL CALL	Call the tool data in the NC program **Further information:** "Tool call by TOOL CALL", Page 277
SPEC FCT	In the **Insert NC function** window, open the **Special functions** folder (e.g., for later programming of a workpiece blank)
PGM CALL	In the **Insert NC function** window, open the **Selection** folder (e.g., to call an external NC program)

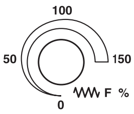
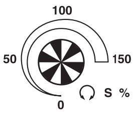
Keycaps for axis input and value input

Key	Function
X ... V	Select axes in the **Manual** operating mode, or enter them in the **Editor** operating mode
0 ... 9	Enter numbers (e.g., coordinate values)
.	Insert a decimal separator during entry
-/+	Invert algebraic sign of entered value
⌫	Delete values during entry
⌖	Open position display of the status overview to copy axis values
Q	In the **Editor** operating mode, within the **Insert NC function** window, open the **FN** folder
CE	Clear entries or delete messages
DEL	Delete NC block or cancel a dialog during programming
NO ENT	Skip or remove optional syntax elements during programming
ENT	Confirm entries and continue dialogs
END	Conclude entry, e.g. finish an NC block
P	Switch between entry of polar and Cartesian coordinates
I	Switch between entry of incremental and absolute coordinates

Keycaps for navigation

Key	Function
↑ . . . →	Position the cursor
GOTO ☐	■ Position the cursor by using the block number of an NC block ■ Open the selection menu while editing
HOME	Jump to first line of an NC program or first column of a table
END	Jump to last line of an NC program or last column of a table
PG UP	Go one page up in an NC program or table
PG DN	Go one page down in an NC program or table
	Mark the active application in order to navigate between applications
	Navigate between areas of an application

Potentiometers

Potentiometer	Function
	Increase or reduce the feed rate **Further information:** "Feed rate F", Page 282
	Increase or reduce the spindle speed **Further information:** "Spindle speed S", Page 281

2.8.3 Icons on the control's user interface

Overview of icons not specific to any operating mode

This overview describes icons that are used in more than one operating mode or that are available regardless of operating mode.

Icons that are specific to individual workspaces are described there.

Icon or shortcut	Function
	Back
	Select the **Home** operating mode
	Select the **Files** operating mode
	Select the **Tables** operating mode
	Select the **Editor** operating mode
	Select the **Manual** operating mode
	Select the **Program Run** operating mode
	Select the **Machine** operating mode
	Open and close the calculator
	Open and close the virtual keyboard
	Open and close the settings
	■ White: Expand control bar or machine manufacturer bar ■ Green: Collapse control bar or machine manufacturer bar or go back ■ Gray: Confirm message
	Add
	Open file
	Close
	Maximize workspace
	Reduce workspace
	■ Black: Add to favorites ■ Yellow: Remove from favorites
CTRL+S	Save
	Save as

Icon or shortcut	Function
CTRL+F	Find
CTRL+C	Copy
CTRL+V	Paste
	Open settings
CTRL+Z	Undo an action
CTRL+Y	Redo an action
	Open selection menu
	Open message menu

2.8.4 Desktop menu workspace

Application

In the **Desktop menu** workspace the control displays selected control and HEROS functions.

Description of function

The **Desktop menu** workspace contains the following areas:

- **Control**

  In this area you can open operating modes or applications.

  Further information: "Overview of operating modes", Page 90

  Further information: "Overview of workspaces", Page 93
- **Tools**

  In this area you can open some tools from the HEROS operating system.

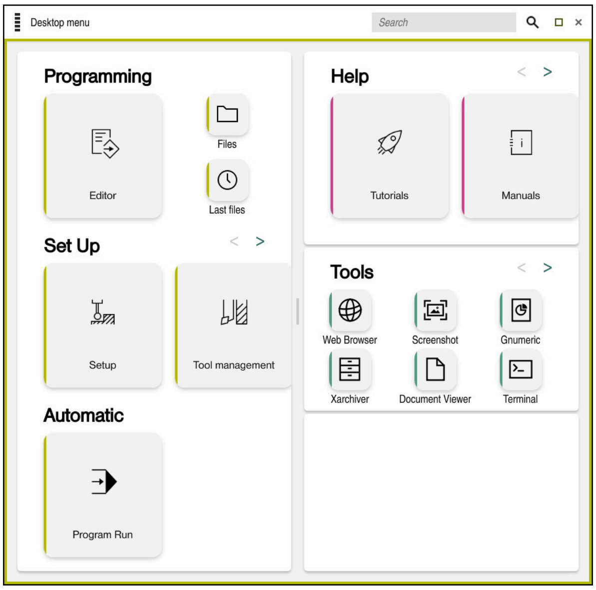
  Further information: "HEROS Operating System", Page 1963
- **Help**

  In this area you can open training videos or TNCguide.
- **Favorites**

  In this area you will find the favorites that you have chosen.

  Further information: "Adding and removing favorites", Page 105

In the title bar you can perform a full-text search for any strings.



Desktop menu workspace

The **Desktop menu** workspace is available in the **Start/Login** application.

Showing or hiding an area

To show or hide an area in the **Desktop menu** workspace:

- ▶ Hold or right-click anywhere within the workspace
- > The control displays a plus sign or minus sign within each area.
- ▶ Select a plus sign
- > The controls shows that area.

> Use the minus sign to hide an area.

Adding and removing favorites

Adding favorites

To add favorites in the **Desktop menu** workspace:

- ▶ Use the full-text search
- ▶ Hold or right-click the function's icon
- > The control displays the icon for **adding favorites**.

- ▶ Select **Add favorite**
- > The control adds the function to the **Favorites** area.

Removing favorites

To remove favorites from the **Desktop menu** workspace:

- ▶ Hold or right-click the function's icon
- > The control displays the icon for **removing favorites**.

- ▶ Select **Remove favorite**
- > The control removes the function from the **Favorites** area.

3

First Steps

3.1 Chapter overview

This chapter uses an example workpiece to explain how to operate the control: from switching the machine on to the finished workpiece.

The chapter covers the following topics:

- Switching the machine on
- Programming and simulating a workpiece
- Setting up tools
- Setting up the workpiece
- Machining the workpiece
- Switching the machine off

3.2 Switching the machine and the control on

Start/Login workspace

DANGER

Caution: hazard to the user!

Machines and machine components always pose mechanical hazards. Electric, magnetic, or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

- Read and follow the machine manual
- Read and follow the safety precautions and safety symbols
- Use the safety devices

Refer to your machine manual.

Switching on the machine and traversing the reference points can vary depending on the machine tool.

To switch the machine and the control on:

- ▶ Switch the power supply of the control and of the machine on
- > The control is in start-up mode and shows the progress in the **Start/Login** workspace.
- > The control displays the **Power interrupted** dialog in the **Start/Login** workspace.

OK

- ▶ Press **OK**
- > The control compiles the PLC program.

I

- ▶ Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit.
- > If the machine is equipped with absolute linear and angle encoders, the control is now ready for operation.
- > If the machine is equipped with incremental linear and angle encoders, the control opens the **Move to ref. point** application.

  Further information: "Referencing workspace", Page 174

- ▶ Press the **NC Start** key
- > The control moves to all necessary reference points.
- > The control is ready for operation and the **Manual operation** application is open.

  Further information: "Manual operation application", Page 180

More detailed information

- Switching on and off

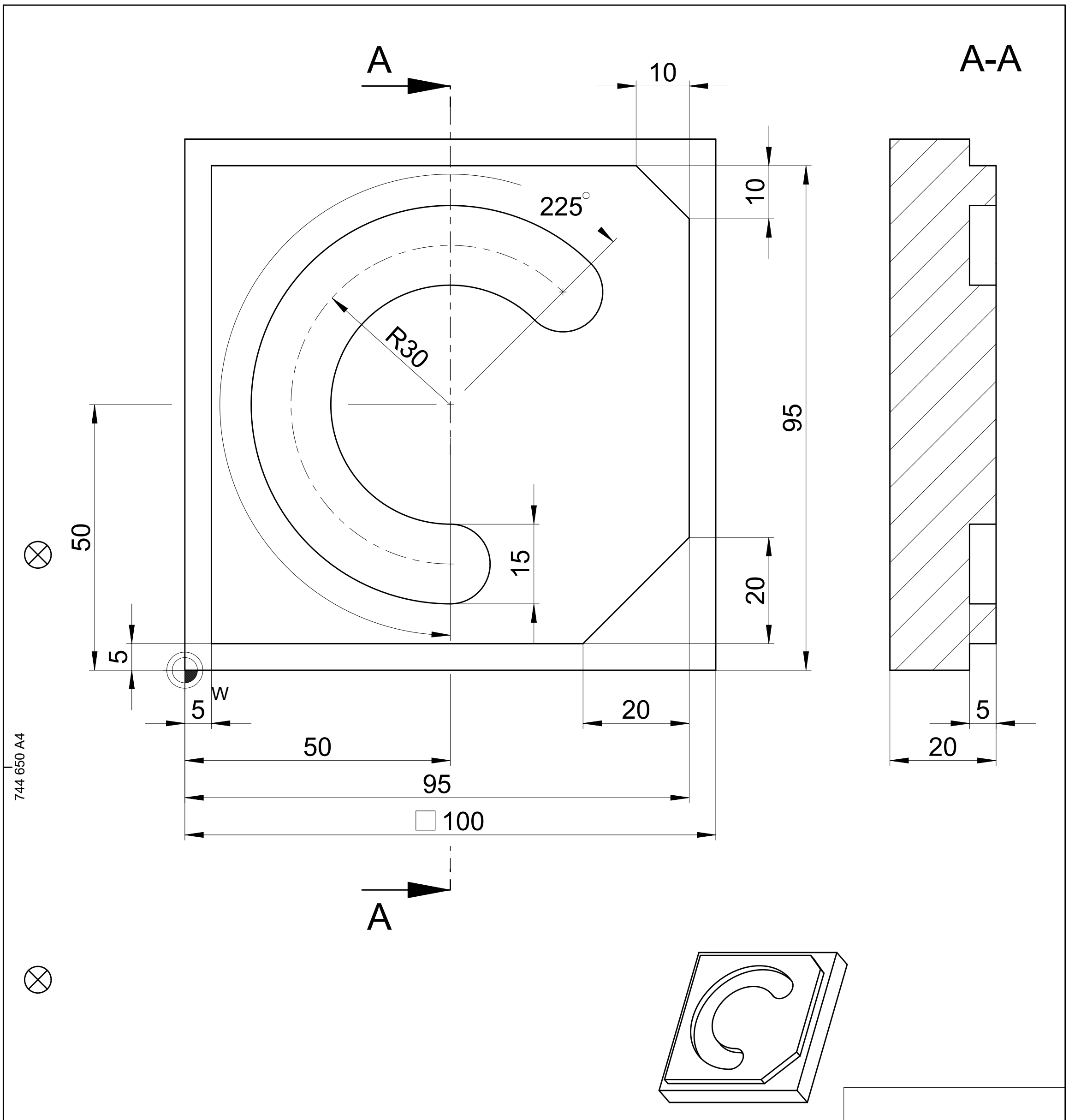
  Further information: "Powering On and Off", Page 171
- Position encoders

  Further information: "Position encoders and reference marks", Page 187
- Axis reference run

  Further information: "Referencing workspace", Page 174

3.3 Programming and simulating a workpiece

3.3.1 Example task 1338459



			ID number
Text:			Change No. C000941-05 Phase: Nicht-Serie
Original drawing Scale 1:1 | Format A4 RoHS	Platte Plate		Werkstoff: Material:
Maße in mm / Dimensions in mm	Einzelteilzeichnung / Component Drawing		●blanke Flächen/Blank surfaces
Werkstückkanten nach ISO 13715 Workpiece edges ISO 13715 -0.3 +0.3	Allgemeintoleranzen ISO 2768-mH ≦6mm: ±0,2 General tolerances ISO 2768-mH ≦6mm: ±0,2	Tolerierung nach ISO 8015 Tolerances as per ISO 8015 Oberflächenbehandlung: Surface treatment:	Oberflächen nach ISO 1302 Surfaces as per ISO 1302

The reproduction, distribution and utilization of this document as well as the communication of its contents to others without express authorization is prohibited. Offenders will be held liable for the payment of damages. All rights reserved in the event of the grant of a patent, utility model or design. (ISO 16016)

HEIDENHAIN	Created	Responsible	Released	Document number	Page
DR. JOHANNES HEIDENHAIN GmbH 83301 Traunreut, Germany	M-TS 05.08.2021			D1358459-00-A-01 (Version Revision Sheet)	1 of 1

3.3.2 Selecting the Editor operating mode

NC programs are always programmed in the **Editor** operating mode.

Requirement

- It must be possible to select the icon of the operating mode

  In order to be able to select the **Editor** operating mode, the control must have already progressed enough during booting that the operating mode icon is no longer dimmed.

Selecting the Editor operating mode

To select the **Editor** operating mode:

- ▶ Select the **Editor** operating mode
- \> The control displays the **Editor** operating mode and the most recently opened NC program.

More detailed information

- Operating mode: **Editor**

  Further information: "Editor operating mode", Page 192

3.3.3 Configuring the control's user interface for programming

The **Editor** operating mode gives you several possibilities for writing an NC program.

> The first steps describe the procedure when you are in the **Klartext programming** mode and the **Form** column is open.

Opening the Form column

You can open the **Form** column only if an NC program is open.

To open the **Form** column:

- ▶ Select **Form**
- \> The control opens the **Form** column

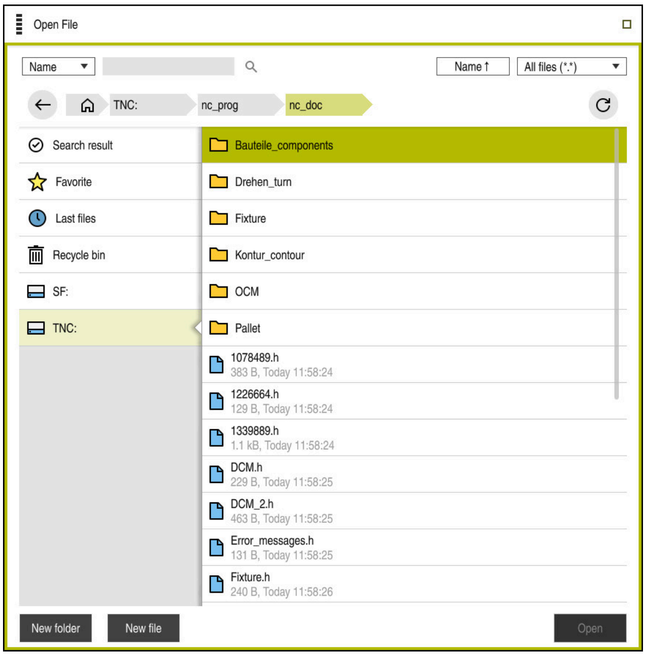
More detailed information

- Editing an NC program

  Further information: "Editing NC programs", Page 202
- Column: **Form**

  Further information: "Form column in the Program workspace", Page 201

3.3.4 Creating a new NC program



Open File workspace in the **Editor** operating mode

To create an NC program in the **Editor** operating mode:

- Select **Add**
- The control displays the **Quick selection** and **Open File** workspaces.

- Select the desired drive in the **Open File** workspace

- Select a folder

Neue Datei

- Select **New file**
- Enter a file name (e.g., 1338459.h)

ENT

- Confirm with the **ENT** key

Open

- Select **Open**
- The control opens a new NC program and the **Insert NC function** window for definition of the workpiece blank.

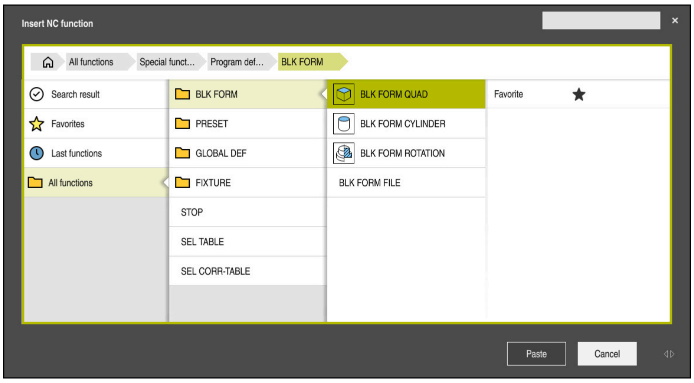
More detailed information

- Workspace: **Open File**
  Further information: "Open File workspace", Page 1075
- Operating mode: **Editor**
  Further information: "Editor operating mode", Page 192

3.3.5 Defining the workpiece blank

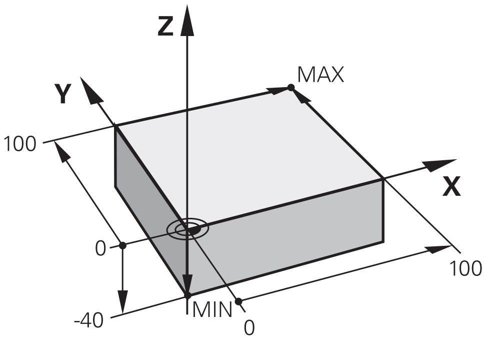
For the NC program you can define a workpiece blank that the control then uses for the simulation. When creating an NC program, the control automatically displays the **Insert NC function** window for definition of the workpiece blank.

> If you close the window without selecting a workpiece blank, you can use the **Insert NC function** button to select the definition of the workpiece blank.



Insert NC function window for defining the workpiece blank

Defining a cuboid workpiece blank



Cuboid workpiece blank with minimum point and maximum point

You define a cuboid through a diagonal in space by entering the minimum point and maximum point relative to the active workpiece preset.

> You can confirm the entries as follows:
> - **ENT** key
> - Right arrow key
> - Click or tap the next syntax element

To define a cuboid workpiece blank:

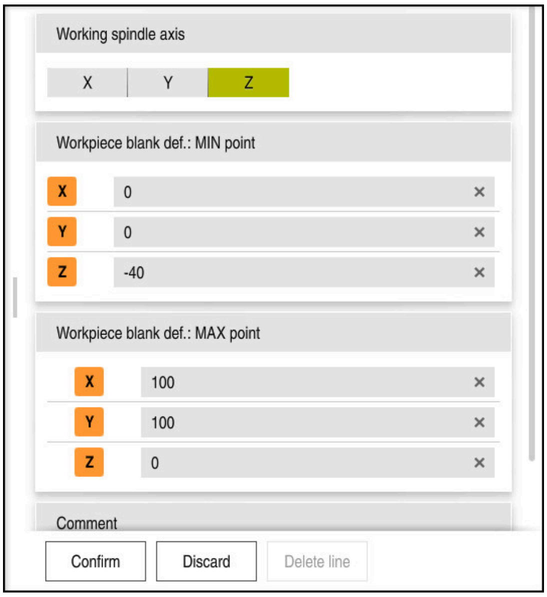
- Select **BLK FORM QUAD**

Paste

- Select **Paste**
- > The control inserts the NC block for definition of the workpiece blank.
- Open the **Form** column
- Select the tool axis (e.g., **Z**)
- Confirm your input
- Enter the smallest X coordinate (e.g., **0**)
- Confirm your input
- Enter the smallest Y coordinate (e.g., **0**)
- Confirm your input
- Enter the smallest Z coordinate (e.g., **-40**)
- Confirm your input
- Enter the largest X coordinate (e.g., **100**)
- Confirm your input
- Enter the largest Y coordinate (e.g., **100**)
- Confirm your input
- Enter the largest Z coordinate (e.g., **0**)
- Confirm your input

Confirm

- Select **Confirm**
- > The control concludes the NC block.



Form column with the defined columns

```
0 BEGIN PGM 1339889 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-40
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 END PGM 1339889 MM
```

More detailed information

- Inserting the workpiece blank
  Further information: "Defining a workpiece blank with BLK FORM", Page 234
- Reference points in the machine
  Further information: "Presets in the machine", Page 187

3.3.6 Structure of an NC program

Using a uniform structure for an NC program offers the following advantages:

- Improved overview
- Quicker programming
- Fewer sources of error

Recommended structure for a contouring program

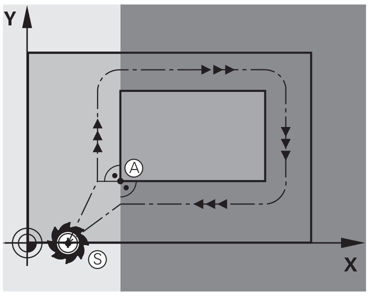
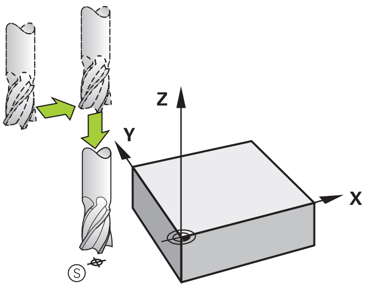
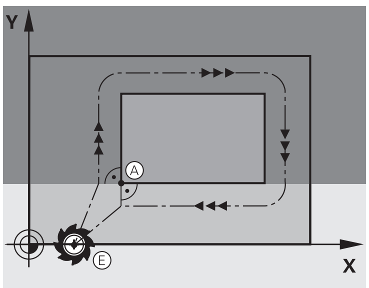
> The control automatically inserts the **BEGIN PGM** and **END PGM** NC blocks.

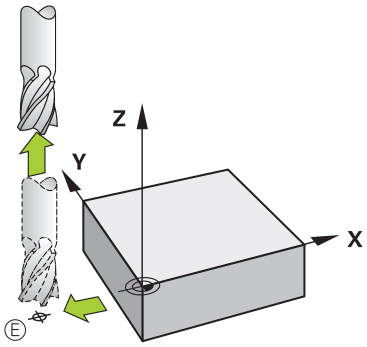
1. **BEGIN PGM** with selection of the unit of measure
2. Define the workpiece blank
3. Call the tool, with the tool axis and the technological data
4. Move the tool to a safe position, and switch the spindle on
5. Pre-position the tool in the working plane, near the first contour point
6. Pre-position the tool in the tool axis, turn coolant on if necessary
7. Approach the contour, activate tool radius compensation if necessary
8. Machine the contour
9. Depart from the contour, turn coolant off
10. Move the tool to a safe position
11. Conclude the NC program
12. **END PGM**

3.3.7 Contour approach and departure

When you program a contour, you need a starting point and end point outside the contour.

The following positions are necessary for contour approach and departure:

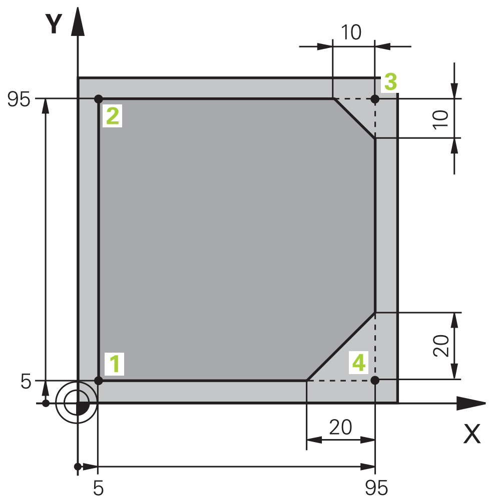
Help graphic	Position
	**Starting point** The following preconditions apply for the starting point: ▪ No tool radius compensation ▪ Approachable without danger of collision ▪ Near to the first contour point The graphic shows the following information: If you define the starting point to be in the dark gray area, the contour will be damaged when the first contour point is approached.
	**Approaching the starting point in the tool axis** Before approaching the first contour point, you must position the tool to the working depth in the tool axis. If there is a danger of collision, approach the starting point in the tool axis separately.
	First contour point The control moves the tool from the starting point to the first contour point. You need to program tool radius compensation for the tool movement to the first contour point.
	**End point** The following preconditions apply for the end point: ▪ Approachable without danger of collision ▪ Near to the last contour point ▪ In order to make sure that the contour will not be damaged, the optimal ending point should lie on the extended tool path for machining the last contour element The graphic shows the following information: If you define the end point to be in the dark gray area, the contour will be damaged when the end point is approached.

Help graphic	Position
	**Departing from the end point in the tool axis** Program the tool axis separately when departing from the end point.
	Identical starting and end points Do not program any tool radius compensation if the starting point and end point are the same. In order to make sure that the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

More detailed information

- Functions for approaching and departing from the contour
  Further information: "Approaching and departing from a contour", Page 317

3.3.8 Programming a simple contour



Workpiece to be programmed

The following texts show you how to mill once at a depth of 5 mm around the contour shown here. You have already defined the workpiece blank.

Further information: "Defining the workpiece blank", Page 113

After you have inserted an NC function, the control shows an explanation about the current syntax element in the dialog bar. You can enter the data directly in the form.

> Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.

Calling a tool

Form column with the syntax elements of the tool call

To call a tool:

TOOL CALL

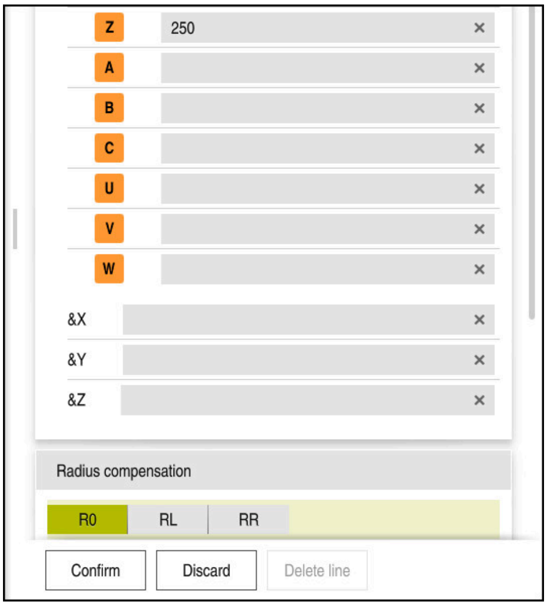
- Select **TOOL CALL**
- Select **Number** in the form
- Enter the tool number (e.g., **16**)
- Select the tool axis **Z**
- Select the spindle speed **S**
- Enter the spindle speed (e.g., **6500**)

Confirm

- Select **Confirm**
- > The control concludes the NC block.

```
3 TOOL CALL 16 Z S6500
```

Moving the tool to a safe position



Form column with the syntax elements of a straight line

To move the tool to a safe position:

- ▶ Select the path function **L**

Z

- ▶ Select **Z**
- ▶ Enter a value (e.g., **250**)
- ▶ Select tool radius compensation **R0**
- > The control applies **R0**, which means there is no tool radius compensation.
- ▶ Select the **FMAX** feed rate
- > The control adopts **FMAX** for rapid traverse.
- ▶ If needed, enter a miscellaneous function **M**, such as **M3** (turn spindle on)

Confirm

- ▶ Select **Confirm**
- > The control concludes the NC block.

```
4 L Z+250 R0 FMAX M3
```

Pre-positioning in the working plane

To pre-position in the working plane:

- ▶ Select the path function **L**

X

- ▶ Select **X**
- ▶ Enter a value (e.g., **-20**)

Y

- ▶ Select **Y**
- ▶ Enter a value (e.g., **-20**)
- ▶ Select the **FMAX** feed rate

Confirm

- ▶ Select **Confirm**
- > The control concludes the NC block.

```
5 L X-20 Y-20 FMAX
```

Pre-positioning in the tool axis

To pre-position in the tool axis:

- Select the path function **L**

Z

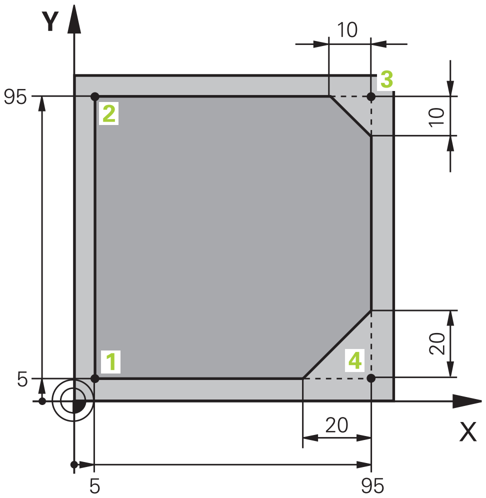
- Select **Z**
- Enter a value (e.g., **-5**)
- Select the feed rate **F**
- Enter the value for the positioning feed rate (e.g., **3000**)
- If needed, enter a miscellaneous function **M**, such as **M8** (turn coolant on)

Confirm

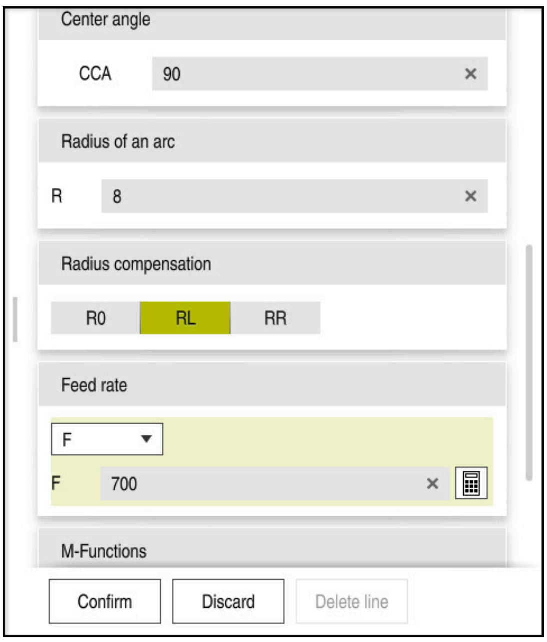
- Select **Confirm**
- > The control concludes the NC block.

```
6  L  Z-5 R0 F3000 M8
```

Approaching the contour



Workpiece to be programmed



Form column with the syntax elements of an approach function

To approach the contour:

APPR /DEP

- ▶ Select the **APPR DEP** path function
- > The control opens the **Insert NC function** window.
- ▶ Select **APPR**
- ▶ Select an approach function (e.g., **APPR CT**)

Paste

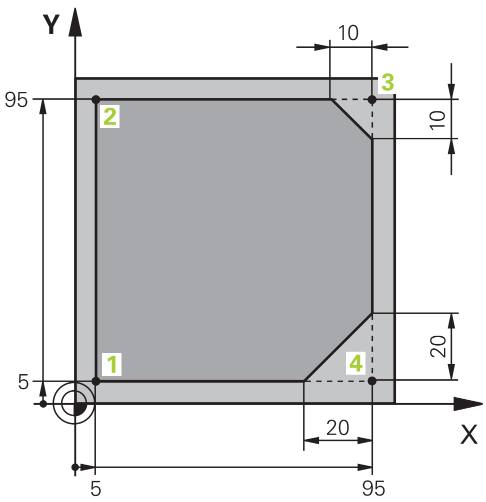
- ▶ Select **Paste**
- ▶ Enter the coordinates of starting point **1** (e.g., **X 5 Y 5**)
- ▶ For the center angle **CCA**, enter the approach angle (e.g., **90**)
- ▶ Enter the radius of the circular arc (e.g., **8**)
- ▶ Select **RL**
- > The control applies tool radius compensation to the left.
- ▶ Select the feed rate **F**
- ▶ Enter the value for the machining feed rate (e.g., **700**)

Confirm

- ▶ Select **Confirm**
- > The control concludes the NC block.

```
7  APPR CT  X+5 Y+5 CCA90 R+8 RL F700
```

Machining a contour



Workpiece to be programmed

To machine the contour:

L
- Select the path function **L**
- Enter the coordinates of contour point 2 that differ (e.g., **Y 95**)

Confirm
- Conclude the NC block with **Confirm**
- > The control applies the changed value and retains all of the other information from the previous NC block.

L
- Select the path function **L**
- Enter the coordinates of contour point 3 that differ (e.g., **X 95**)

Confirm
- Conclude the NC block with **Confirm**

CHF
- Select the path function **CHF**
- Enter the chamfer width (e.g., **10**)

Confirm
- Conclude the NC block with **Confirm**

L
- Select the path function **L**
- Enter the coordinates of contour point 4 that differ (e.g., **Y 5**)

Confirm
- Conclude the NC block with **Confirm**

CHF
- Select the path function **CHF**
- Enter the chamfer width (e.g., **20**)

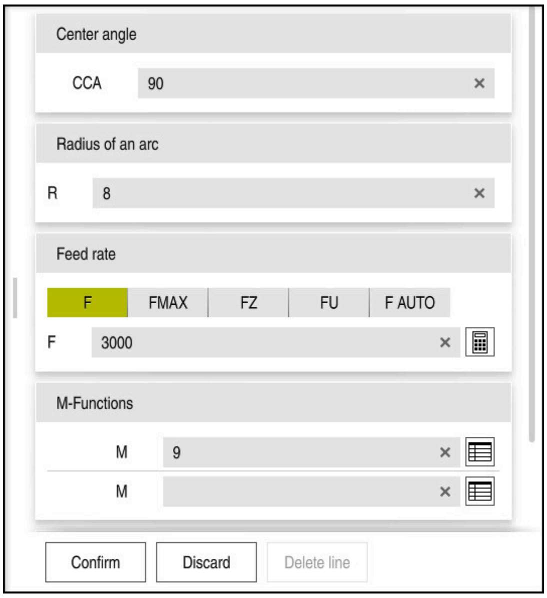
Confirm
- Conclude the NC block with **Confirm**

L
- Select the path function **L**
- Enter the coordinates of contour point 1 that differ (e.g., **X 5**)

Confirm
- Conclude the NC block with **Confirm**

```
8 L Y+95
9 L X+95
10 CHF 10
11 L Y+5
12 CHF 20
13 L X+5
```

Departing from the contour



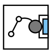
Form column with the syntax elements of a departure function

To depart from the contour:

APPR /DEP

- Select the **APPR DEP** path function
- > The control opens the **Insert NC function** window.

- Select **DEP**



- Select a departure function (e.g., **DEP CT**)

Paste

- Select **Paste**
- For the center angle **CCA**, enter the departure angle (e.g., **90**)
- Enter the departure radius (e.g., **8**)
- Select the feed rate **F**
- Enter the value for the positioning feed rate (e.g., **3000**)
- If needed, enter a miscellaneous function **M**, such as **M9** (turn coolant off)

Confirm

- Select **Confirm**
- > The control concludes the NC block.

```
14  DEP CT  CCA90 R+8 F3000 M9
```

Moving the tool to a safe position

To move the tool to a safe position:

- Select the path function **L**
- Select **Z**
- Enter a value (e.g., **250**)
- Select tool radius compensation **R0**
- Select the **FMAX** feed rate
- Enter a miscellaneous function **M** if required
- Select **Confirm**
- > The control concludes the NC block.

```
15  L  Z+250 R0 FMAX M30
```

More detailed information

- Tool call
  Further information: "Tool call by TOOL CALL", Page 277
- Line **L**
  Further information: "Straight line L", Page 297
- Designation of the axes and the working plane
  Further information: "Designation of the axes on milling machines", Page 186
- Functions for approaching and departing from the contour
  Further information: "Approaching and departing from a contour", Page 317
- Chamfer **CHF**
  Further information: "Chamfer CHF", Page 297
- Miscellaneous functions
  Further information: "Overview of miscellaneous functions", Page 1225

3.3.9 Programming a machining cycle

The following texts show you how to mill the circular slot of the example task at a depth of 5 mm. You have already defined the workpiece blank and created the outside contour.

Further information: "Example task 1338459", Page 110

After you have inserted a cycle, you can define the associated values in the cycle parameters. You can program the cycle directly in the **Form** column.

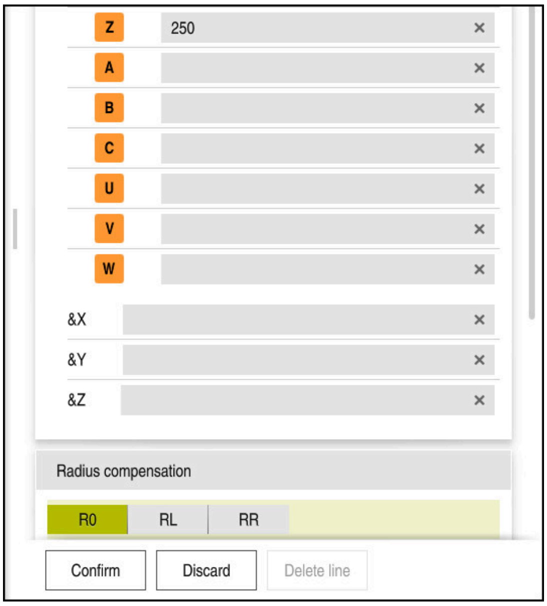
Calling a tool

To call a tool:

- Select **TOOL CALL**
- Select **Number** in the form
- Enter the tool number (e.g., **6**)
- Select the tool axis **Z**
- Select the spindle speed **S**
- Enter the spindle speed (e.g., **6500**)
- Select **Confirm**
- > The control concludes the NC block.

```
16  TOOL CALL 6 Z S6500
```

Move the tool to a safe position



Form column with the syntax elements of a straight line

To move the tool to a safe position:

- Select the path function **L**

Z

- Select **Z**
- Enter a value (e.g., **250**)
- Select tool radius compensation **R0**
- > The control applies **R0**, which means there is no tool radius compensation.
- Select the **FMAX** feed rate
- > The control adopts **FMAX** for rapid traverse.
- If needed, enter a miscellaneous function **M**, such as **M3** (turn spindle on)

Confirm

- Select **Confirm**
- > The control concludes the NC block.

```
17 L  Z+250 R0 FMAX M3
```

Pre-positioning in the working plane

To pre-position in the working plane:

- Select the path function **L**

X

- Select **X**
- Enter a value (e.g., **+50**)

Y

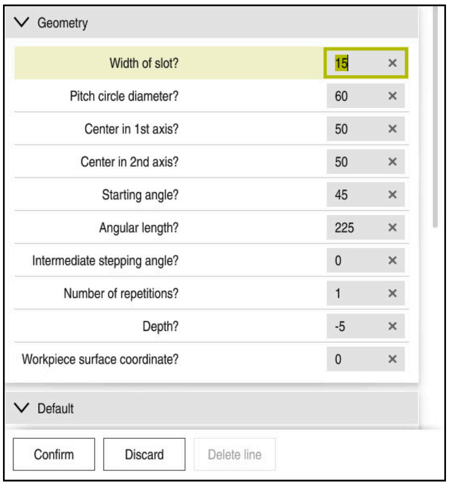
- Select **Y**
- Enter a value (e.g., **+50**)
- Select the **FMAX** feed rate

Confirm

- Select **Confirm**
- > The control concludes the NC block.

```
18 L  X+50 Y+50 FMAX
```

Defining a cycle



Form column with possibilities for entering cycle information

To define the circular slot:

CYCL DEF

- Select the **CYCL DEF** key
- > The control opens the **Insert NC function** window.

CYCL DEF

- Select Cycle **254 CIRCULAR SLOT**
- > The control initiates a dialog and prompts you for all required input values.
- Enter all input values in the form

Paste

- Select **Paste**
- > The control initiates a dialog and prompts you for all required input values.
- Enter all input values in the form

Confirm

- Select **Confirm**
- > The control saves the cycle.

```
19 CYCL DEF 254 CIRCULAR SLOT ~
    Q215=+0            ;MACHINING OPERATION ~
    Q219=+15           ;SLOT WIDTH ~
    Q368=+0.1          ;ALLOWANCE FOR SIDE ~
    Q375=+60           ;PITCH CIRCLE DIAMETR ~
    Q367=+0            ;REF. SLOT POSITION ~
    Q216=+50           ;CENTER IN 1ST AXIS ~
    Q217=+50           ;CENTER IN 2ND AXIS ~
    Q376=+45           ;STARTING ANGLE ~
    Q248=+225          ;ANGULAR LENGTH ~
    Q378=+0            ;STEPPING ANGLE ~
    Q377=+1            ;NR OF REPETITIONS ~
    Q207=+500          ;FEED RATE MILLING ~
    Q351=+1            ;CLIMB OR UP-CUT ~
    Q201=-5            ;DEPTH ~
    Q202=+5            ;PLUNGING DEPTH ~
    Q369=+0.1          ;ALLOWANCE FOR FLOOR ~
    Q206=+150          ;FEED RATE FOR PLNGNG ~
    Q338=+5            ;INFEED FOR FINISHING ~
    Q200=+2            ;SET-UP CLEARANCE ~
    Q203=+0            ;SURFACE COORDINATE ~
    Q204=+50           ;2ND SET-UP CLEARANCE ~
    Q366=+2            ;PLUNGE ~
    Q385=+500          ;FINISHING FEED RATE ~
    Q439=+0            ;FEED RATE REFERENCE
```

Calling a cycle

To call the cycle:

CYCL CALL

- Select **CYCL CALL**

```
20 CYCL CALL
```

Moving the tool to a safe position and concluding the NC program

To move the tool to a safe position:

L

- Select the path function **L**

Z

- Select **Z**
- Enter a value (e.g., **250**)
- Select tool radius compensation **R0**
- Select the **FMAX** feed rate
- Enter a miscellaneous function **M**, such as **M30** (program end)

Confirm

- Select **Confirm**
- > The control concludes the NC block and the NC program.

```
21  L  Z+250 R0 FMAX M30
```

More detailed information

- Machining cycles
  Further information: "Machining Cycles", Page 419
- Calling a cycle
  Further information: "Calling cycles", Page 425

3.3.10 Configuring the control's user interface for simulation

In the **Editor** operating mode you can test NC programs graphically. The control simulates the active NC program in the **Program** workspace.

In order to simulate the NC program you must open the **Simulation** workspace.

> For the simulation you can close the **Form** column to get a better view of the NC program and the **Simulation** workspace.

Opening the Simulation workspace

You can open additional workspaces in the **Editor** operating mode only if an NC program is open.

To open the **Simulation** workspace:

- ▶ In the application bar, select **Workspaces**
- ▶ Select **Simulation**
- \> The control then additionally displays the **Simulation** workspace.

> You can also open the **Simulation** workspace with the **Test Run** operating mode key.

Configuring the Simulation workspace

You can simulate the NC program without needing to enter any special settings. However, an adjustment to the simulation speed is recommended for best viewing of the simulation.

To adjust the speed of the simulation:

- ▶ Use the slider to select the factor (e.g., **5.0 * T**)
- \> The control then performs the subsequent simulation at five times the speed of the programmed feed rate.

If you use different tables, such as tool tables, for program run and the simulation, then you can define the tables in the **Simulation** workspace.

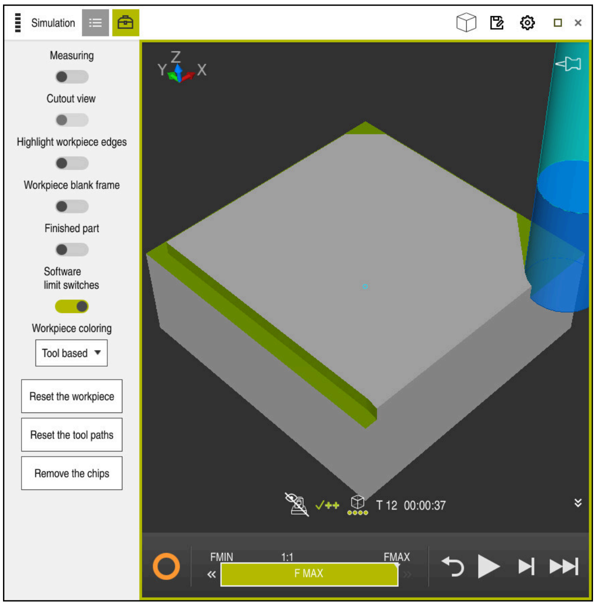
More detailed information

- Workspace: **Simulation**
  Further information: "Simulation Workspace", Page 1405

3.3.11 Simulating an NC program

You test the NC program in the **Simulation** workspace.

Starting the simulation



Simulation workspace in the **Editor** operating mode

You start the simulation as follows:

- ▶ Select **Start**
- > The control asks whether the file should be saved.

- ▶ Select **Save**
- > The control starts the simulation.
- > The control uses the **Control-in-operation** symbol to show the simulation status.

Definition

Control-in-operation:
The control uses the **Control-in-operation** symbol to show the current simulation status in the action bar and on the tab of the NC program:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

More detailed information

- **Simulation** workspace
  Further information: "Simulation Workspace", Page 1405

3.4 Configuring a tool

3.4.1 Selecting the Tables operating mode

You configure tools in the **Tables** operating mode.

To select the **Tables** operating mode:

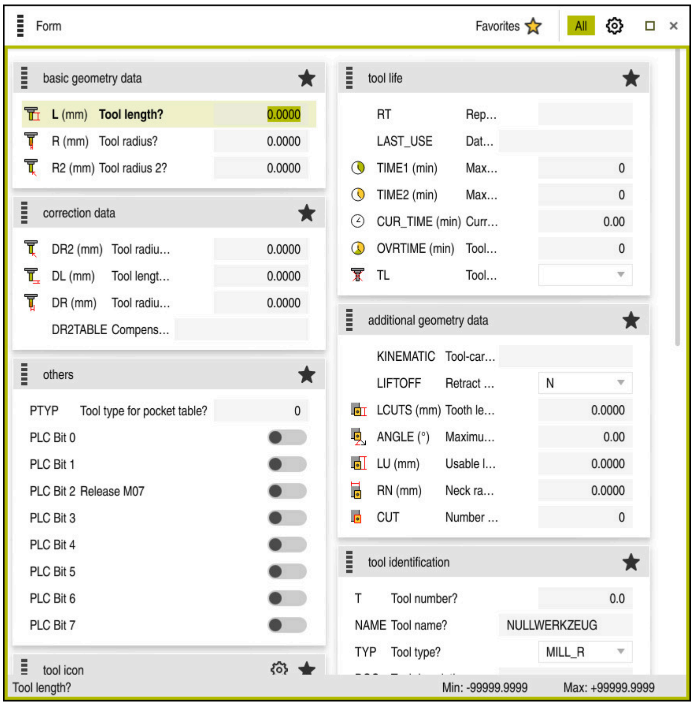
- ▶ Select the **Tables** operating mode
- > The control displays the **Tables** operating mode.

More detailed information

- Operating mode: **Tables**

  Further information: "Tables operating mode", Page 1802

3.4.2 Configuring the control's user interface



Form workspace in the **Tables** operating mode

In the **Tables** operating mode you open and edit the various tables of the control either in the **Table** workspace or in the **Form** workspace.

> ℹ The first steps describe the procedure with the **Form** workspace open.

To open the **Form** workspace:

- ▶ In the application bar, select **Workspaces**
- ▶ Select **Form**
- > The control opens the **Form** workspace.

More detailed information

- **Form** workspace

  Further information: "Form workspace for tables", Page 1807
- Workspace: **Table**

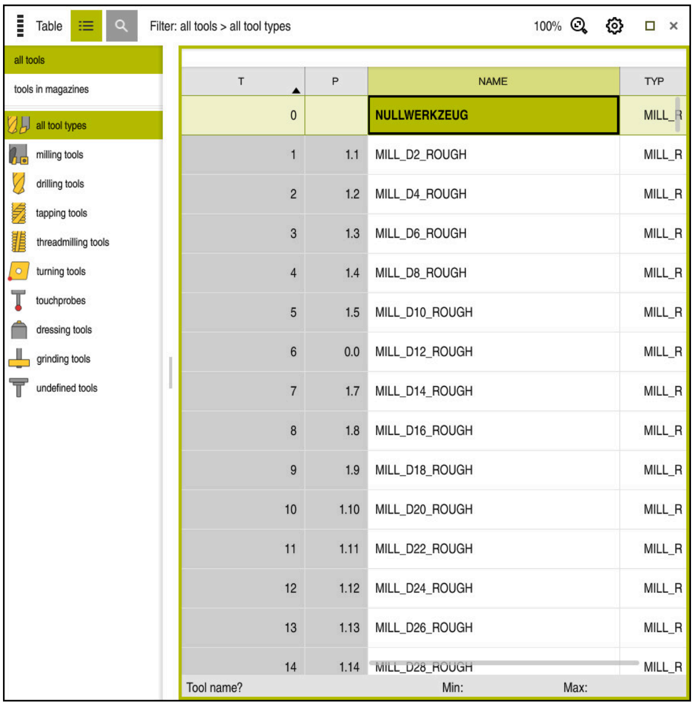
  Further information: "Table workspace", Page 1803

3.4.3 Preparing and measuring tools

To prepare tools:

- ▶ Clamp the required tools in their tool holders
- ▶ Measure the tools
- ▶ Write down the length and the radius or transfer these directly to the control

3.4.4 Editing within tool management



Tool management application in the **Table** workspace

Tool management allows you to save tool data, such as the length and radius as well as other tool-specific information.
The control displays the tool data for all tool types in tool management. In the **Form** workspace the control displays only the relevant tool data for the current tool type.

To enter the tool data in tool management:

- ▶ Select **Tool management**
- > The control displays the **Tool management** application.
- ▶ Open the **Form** workspace

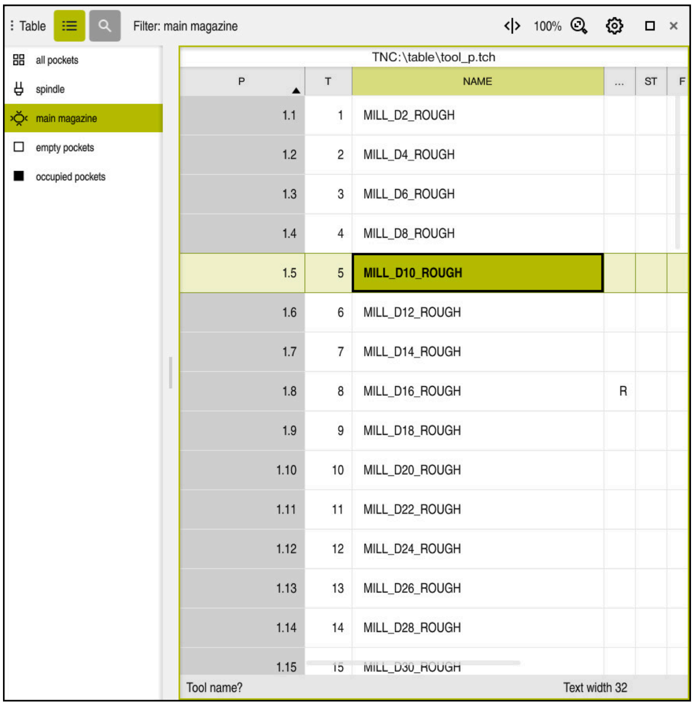
- ▶ Activate **Edit**
- ▶ Select the desired tool number (e.g., **16**)
- > The control displays the tool data of the selected tool in the form.
- ▶ Define the required tool data in the form; for example, the length **L** and the tool radius **R**

More detailed information

- Operating mode: **Tables**
  Further information: "Tables operating mode", Page 1802
- Workspace: **Form**
  Further information: "Form workspace for tables", Page 1807
- Tool management
  Further information: "Tool management ", Page 270
- Tool types
  Further information: "Tool types", Page 255

3.4.5 Editing the pocket table

> Refer to your machine manual!
> Access to the **tool_p.tch** pocket table is machine-dependent.



Pocket table application in the **Table** workspace

The control assigns a pocket in the tool magazine to each tool that is in the tool table. This assignment, as well as the load situation of each tool, is shown in the pocket table.

There are various ways of accessing the pocket table:

- Functions of the machine manufacturer
- Third-party tool-management system
- Manual access to the control

To enter the data in the pocket table:

- ▶ Select **Pocket table**
- > The control displays the **Pocket table** application.
- ▶ Open the **Form** workspace

- ▶ Activate **Edit**
- ▶ Select the desired pocket number
- ▶ Define the tool number
- ▶ Define any additional tool data if necessary, such as whether the pocket is reserved

More detailed information

- Pocket table

  Further information: "Pocket table tool_p.tch", Page 1843

3.5 Setting up a workpiece

3.5.1 Selecting an operating mode

You set up workpieces in the **Manual** operating mode.

To select the **Manual** operating mode:

- Select the **Manual** operating mode
- > The control displays the **Manual** operating mode.

More detailed information

- Operating mode: **Manual**
  Further information: "Overview of operating modes", Page 90

3.5.2 Clamping the workpiece

Mount the workpiece with a fixture on the machine table.

3.5.3 Workpiece presetting with a touch probe

Inserting a workpiece touch probe

Use a workpiece touch probe to set up the workpiece with the aid of the control and set the workpiece preset.

To insert a workpiece touch probe:

T

- Select **T**
- Enter the tool number of the workpiece touch probe, (e.g., **600**)
- Press the **NC Start** key
- > The controls inserts the workpiece touch probe.

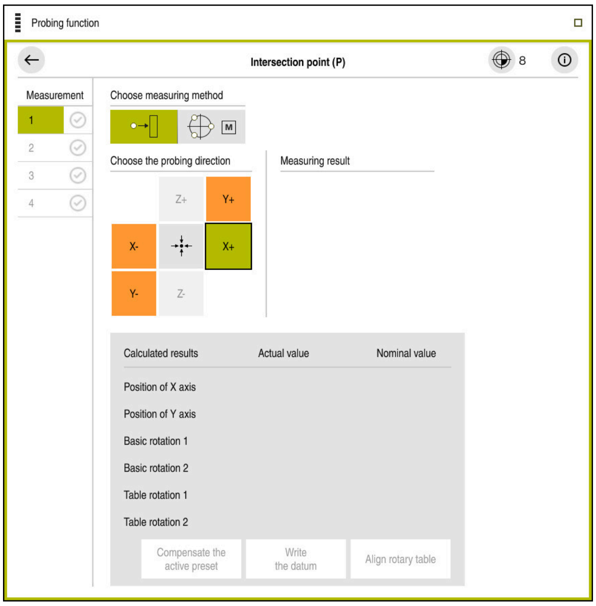
Setting a workpiece preset

To set a workpiece preset at a corner:

- Select the **Setup** application
- Select **Intersection point (P)**
- The control opens the probing cycle.
- Manually position the touch probe near the first touch point of the first workpiece edge
- In the **Choose the probing direction** area, select the direction of probing (e.g., **Y+**)
- Press the **NC Start** key
- The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the second touch point of the first workpiece edge
- Press the **NC Start** key
- The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the first touch point of the second workpiece edge
- In the **Choose the probing direction** area, select the direction of probing (e.g., **X+**)
- Press the **NC Start** key
- The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- Manually position the touch probe near the second touch point of the second workpiece edge
- Press the **NC Start** key
- The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- The control then displays the coordinates of the determined corner point in the **Measuring result** area.

Compensate the active preset

- Select **Compensate the active preset**
- The control applies the calculated results to the workpiece preset.
- Select **Exit probing**
- The control closes the probing cycle.



Probing function workspace with an open manual probing function

More detailed information

- Workspace: **Probing function**

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1425
- Reference points in the machine

  Further information: "Presets in the machine", Page 187
- Tool change in the **Manual operation** application

  Further information: "Manual operation application", Page 180

3.6 Machining a workpiece

3.6.1 Selecting an operating mode

You machine workpieces in the **Program Run** operating mode.

To select the **Program Run** operating mode:

- ▶ Select the **Program Run** operating mode
- > The control displays the **Program Run** operating mode and the most recently executed NC program.

More detailed information

- Operating mode: **Program Run**

  Further information: "Program Run operating mode", Page 1778

3.6.2 Opening an NC program

To open an NC program:

- Select **Open File**
- > The control displays the **Open File** workspace.
- Select an NC program

Open

- Select **Open**
- > The control opens the NC program.

More detailed information

- Workspace: **Open File**

  Further information: "Open File workspace", Page 1075

3.6.3 Starting an NC program

To start an NC program:

- Press the **NC Start** key
- > The control runs the active NC program.

3.7 Switching the machine off

Refer to your machine manual.
Switching off is a machine-dependent function.

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss no matter what state the control was in!

- Always shut down the control
- Only operate the main switch after being prompted on the screen

To shut down the control:

- Select the **Home** operating mode

Shut down

- Select **Shut down**
- > The control opens the **Shut down** window.

Shut down

- Select **Shut down**
- > The control shuts down.
- > Once shutdown has concluded, the control displays the text **Now you can switch off.**

4

Status Displays

4.1 Application

The control shows the status or values of individual functions in the status displays.

The control offer the following status displays:

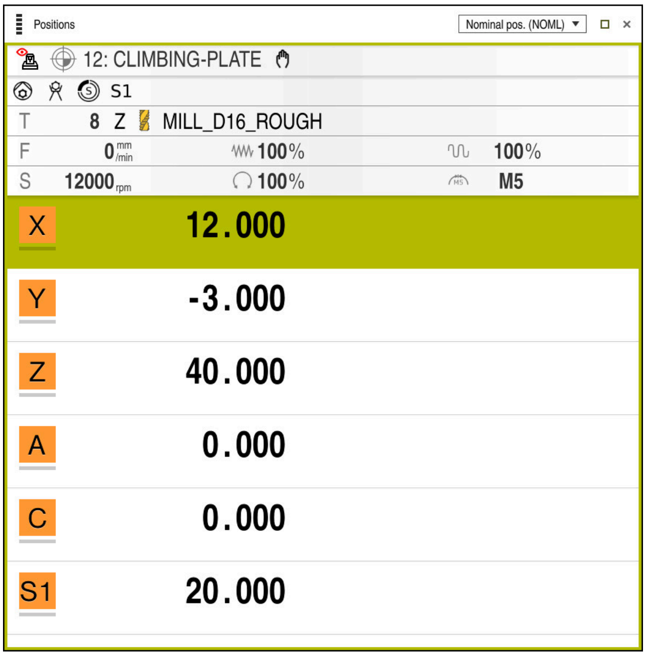
- General status display and position display in the **Positions** workspace
  Further information: "Positions workspace", Page 141
- Status overview on the control bar
  Further information: "Status overview on the control bar", Page 147
- Additional status displays for specific areas in the **Status** workspace
  Further information: "Status workspace", Page 149
- Additional status displays in the **Editor** operating mode in the **Simulation status** workspace, based on the machining status of the simulated workpiece
  Further information: "Simulation status workspace", Page 165

4.2 Positions workspace

Application

The general status display in the **Positions** workspace provides information about the status of various functions of the control and about current axis positions.

Description of function



Positions workspace with general status display

You can open the **Positions** workspace in the following operating modes:

- **Manual**
- **Program Run**

Further information: "Overview of operating modes", Page 90

The **Positions** workspace provides the following information:

- Symbols of active and inactive functions, such as Dynamic Collision Monitoring (DCM, option 40)
- Active tool
- Technology values
- Settings of the spindle and feed-rate potentiometers
- Active miscellaneous functions for the spindle
- Axis values and statuses, such as "Axis not referenced"

  Further information: "Test status of the axes", Page 1908

Axis display and position display

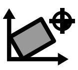
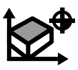
	Refer to your machine manual. In the machine parameter **axisDisplay** (no. 100810) you define the quantity and sequence of the displayed axes.

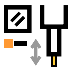
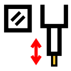
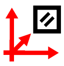
Icon	Meaning
ACTL	Position display mode (e.g., actual or nominal coordinates of the current tool position) You can select the mode in the title bar of the workspace. **Further information:** "Position displays", Page 167
X m	Axes The X axis is selected. You can move the selected axis. The auxiliary axis **m** is not selected. The control displays auxiliary axes, such as the tool magazine, as lowercase letters. **Further information:** "Definition", Page 146
?	The axis is not referenced.
	The axis is not in safe mode. **Further information:** "Checking axis positions manually", Page 1909
Δ	The axis is moving the distance-to-go shown next to the symbol.
	The axis is clamped.
	You can move the axis with the handwheel.
F_{SOS} 0 mm/min	Feed rate when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 1905
S_{SOS} 0 U/min	Spindle when stopped **Further information:** "Functional safety (FS) in the Positions workspace", Page 1905

Presets and technology values

Symbol	Meaning
	Number of the active workpiece preset The number corresponds to the active line number of the preset table. **Further information:** "Preset management", Page 949
T	In the **T** area, the control shows the following information: ■ Number of the active tool ■ Tool axis of the active tool ■ Symbol of the defined tool type ■ Name of the active tool
F	In the **F** area, the control shows the following information: ■ Active feed rate in mm/min You can program the feed rate in various units of measurement. The control always converts the programmed feed rate in this display to mm/min. ■ Setting of the rapid-traverse potentiometer in percent ■ Setting of the feed-rate potentiometer in percent **Further information:** "Potentiometers", Page 101
S	In the **S** area, the control shows the following information: ■ Active shaft speed in rpm If you have programmed a cutting speed instead of a rotational speed, the control automatically converts this value to a rotational speed. ■ Setting of the spindle potentiometer in percent ■ Active miscellaneous function for the spindle

Active functions

Icon	Meaning
	The **Manual traverse** function is active.
	The **Manual traverse** function is not active. **Further information:** "Program Run operating mode", Page 1778
	**RL** tool radius compensation is active. **Further information:** "Tool radius compensation", Page 1038
	**RR** tool radius compensation is active. **Further information:** "Tool radius compensation", Page 1038 These icons are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 1786
	**R+** tool radius compensation is active. **Further information:** "Tool radius compensation", Page 1038
	**R-** tool radius compensation is active. **Further information:** "Tool radius compensation", Page 1038 These icons are transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 1786
	3D tool compensation is active. **Further information:** "3D tool compensation (option 9)", Page 1049 This icon is transparent while the **Block scan** function of the control is active. **Further information:** "Block scan for mid-program startup", Page 1786
	A basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 951
	The basic rotation will be taken into account while moving the axes. **Further information:** "Basic rotation setting", Page 1025
	A 3D basic rotation is defined in the active preset. **Further information:** "Basic rotation and 3D basic rotation", Page 951
	The tilted working plane will be taken into account while moving the axes. **Further information:** "Tilting the working plane with PLANE functions (option 8)", Page 979 **Further information:** "3D ROT setting", Page 1024

Icon	Meaning
	The **Tool axis** function is active. **Further information:** "Tool axis setting", Page 1025
	Either the **TRANS MIRROR** function or Cycle **8 MIRRORING** is active. The axes programmed in the function or cycle are mirrored and moved. **Further information:** "Cycle 8 MIRRORING", Page 959 **Further information:** "Mirroring with TRANS MIRROR", Page 973
	The pulsing spindle speed function **S-PULSE** is active. **Further information:** "Pulsing spindle speed with FUNCTION S-PULSE", Page 1122
	The **PARAXCOMP DISPLAY** function is active.
	The **PARAXCOMP MOVE** function is active. **Further information:** "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1195
	The **PARAXMODE** function is active. This icon might be superimposed on the icons for **PARAXCOMP DISPLAY** and **PARAXCOMP MOVE**. **Further information:** "Select three linear axes for machining with FUNCTION PARAXMODE", Page 1197
TCPM	Either the **M128** function or **TCPM FUNCTION** (option 9) is active. **Further information:** "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027
	The turning mode **FUNCTION MODE TURN** (option 50) is active. **Further information:** "Switching the operating mode with FUNCTION MODE", Page 210
	The grinding mode **FUNCTION MODE GRIND** (option 156) is active. **Further information:** "Switching the operating mode with FUNCTION MODE", Page 210
	Dressing mode (option 156) is active. **Further information:** "Activating dressing mode with FUNCTION DRESS", Page 228
	Dynamic Collision Monitoring (DCM, option 40) is active.
	Dynamic Collision Monitoring (DCM, option 40) is not active. **Further information:** "Dynamic Collision Monitoring (DCM, option 40)", Page 1084

Icon	Meaning
AFC L	Adaptive Feed Control (AFC, option 45) is active in teach-in cut mode.
AFC	Adaptive Feed Control (AFC, option 45) is active in closed-loop mode. **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 1114
ACC	Active Chatter Control (ACC, option 145) is active. **Further information:** "Active Chatter Control (ACC, option 145)", Page 1121
	Global Program Settings (GPS, option 44) are active. **Further information:** "Global Program Settings (GPS, option 44)", Page 1132
	Process Monitoring (option 168) is active. **Further information:** "Process Monitoring (option 168)", Page 1152

In the optional machine parameter **iconPrioList** (no. 100813) you can change the sequence in which the control displays these symbols. The symbol for Dynamic Collision Monitoring (DCM, option 40) is always visible and cannot be configured.

Definition

Auxiliary axes
Auxiliary axes are controlled by the PLC and are not included in the kinematics description. Auxiliary axes are driven, for example, hydraulically, electrically, or by an external motor. The machine manufacturer can define the tool magazine, for example, as an auxiliary axis.

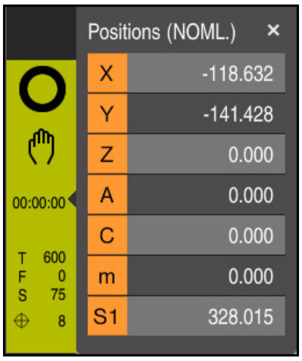
4.3 Status overview on the control bar

Application

On the control bar the control shows a status overview with the machining status, the current technology values, and the axis positions.

Description of function

General information



When you are machining an NC program or individual NC blocks, the control provides the following information on the control bar:

- **Control-in-operation**: current machining status

  Further information: "Definition", Page 148
- Symbol of the application used for machining
- Program run time

  > The control displays the same value for the program run time as on the **PGM** tab of the **Status** workspace.
  >
  > **Further information:** "Display of the program run time", Page 166

- Active tool
- Active feed rate
- Current spindle speed
- Active workpiece preset

Position display

If you select the status overview area, the control opens or closes the position display with the current axis positions. The control uses the same position display mode as in the **Positions** workspace, for example **Actual pos. (ACT)**.

Further information: "Positions workspace", Page 141

If you select an axis line, the control copies the current value of this line to the clipboard.

Press the **actual-position-capture** key to open the position display. The control prompts you to select the value to be copied to the clipboard.

Definition

Control-in-operation:
The control uses the **Control-in-operation** symbol to show the machining status of the NC program or NC block:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

Further information: "Interrupting, stopping or canceling program run", Page 1782

When the control bar is expanded, the control shows additional information about the current status, such as **Active, feed rate at zero**.

4.4 Status workspace

Application

In the **Status** workspace the control shows the additional status display. The additional status display shows the current status of various functions on specific tabs. You can use the additional status display to better monitor the running of an NC program by receiving real-time information about active functions and accesses.

Description of function

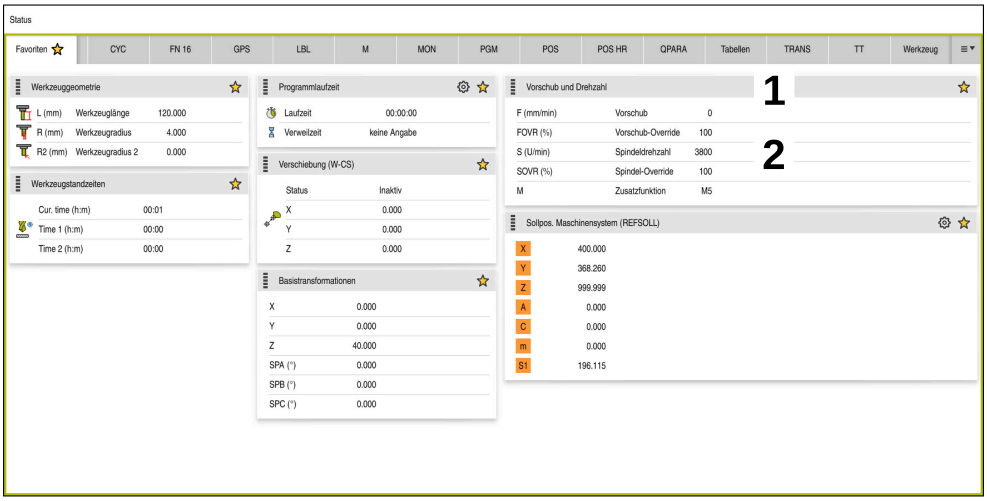
You can open the **Status** workspace in the following operating modes:

- **Manual**
- **Program Run**

Further information: "Overview of operating modes", Page 90

Favorites tab

On the **Favorites** tab you can arrange your own status display with contents from the other tabs.



Favorites tab

1 Area
2 Contents

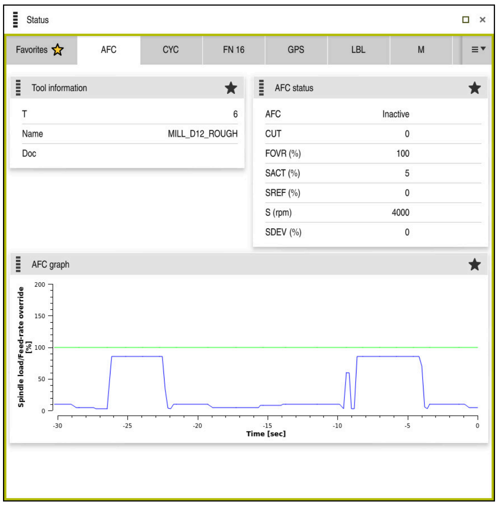
Every area in the status display has a **Favorites** icon. If you select the icon, the control adds that area to the **Favorites** tab.

Further information: "Icons on the control's user interface", Page 102

AFC tab (option 45)

The control displays information about the Adaptive Feed Control function (AFC, option 45) on the **AFC** tab.

Further information: "Adaptive Feed Control (AFC, option 45)", Page 1114



AFC tab

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Doc** Comment about the tool from the tool management
AFC status	■ **AFC** If AFC is being used to control the feed rate, then **Control** is displayed in this area. If the control is not controlling the feed rate, then **Inactive** is displayed in this area. ■ **CUT** Counts the quantity of cuts that have been performed with **FUNCTION AFC CUT BEGIN**, starting from zero. ■ **FOVR (%)** Active factor of the feed-rate potentiometer in percent ■ **SACT (%)** Current spindle load in percent ■ **SREF (%)** Reference load of the spindle in percent Define the reference load of the spindle in the syntax element **LOAD** of the **FUNCTION AFC CUT BEGIN** function. **Further information:** "NC functions for AFC (option 45)", Page 1116 ■ **S (rpm)** Spindle shaft speed in rpm ■ **SDEV (%)** Current deviation of the speed in percent

Area	Contents
AFC graph	The **AFC graph** visualizes the relationship between the elapsed **Time [sec]** and the **Spindle load/Feed-rate override [%]**. The green line in the graph shows the feed-rate override and the blue line shows the spindle load.

CYC tab

On the **CYC** tab the control shows information about machining cycles.

Area	Contents
Active cycle definition	When you use the **CYCLE DEF** function to define a cycle, the control shows the cycle number in this area.
Cycle 32 TOLERANCE	■ **Status** Shows whether Cycle **32 TOLERANCE** is active or inactive ■ Values of Cycle **32 TOLERANCE** ■ Values from the machine manufacturer for path and angle tolerance, such as predefined machine-specific roughing or finishing filters ■ Values of Cycle **32 TOLERANCE** that are restricted by Dynamic Collision Monitoring (DCM, option 40)

The machine manufacturer uses Dynamic Collision Monitoring (DCM, option 40) to define the restriction of the tolerance.

In the optional machine parameter **maxLinearTolerance** (no. 205305) the machine manufacturer defines the maximum permissible linear tolerance. In the optional machine parameter **maxAngleTolerance** (no. 205303) the machine manufacturer defines the maximum permissible angle tolerance. If DCM is active, the control restricts the tolerance defined in **32 TOLERANCE** to these values.

If the tolerance is restricted by DCM, the control displays a gray warning triangle as well as the restricted values.

FN16 tab

On the **FN16** tab the control displays the contents of a file that was output with **FN 16: F-PRINT**.

Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1285

Area	Contents
Output	Contents of an output file that was output with **FN 16: F-PRINT**, such as measured values or texts.

GPS tab (option 44)

The control displays information about the Global Program Settings (GPS, option 44) on the **GPS** tab.

Further information: "Global Program Settings (GPS, option 44)", Page 1132

Area	Contents
Additive offset (M-CS)	■ **Status** The **Status** shows whether a function is active or inactive. A function can be active even if its values are zero. ■ **A (°)** **Additive offset (M-CS)** in the A axis The **Additive offset (M-CS)** function is also available for the other rotary axes **B (°)** and **C (°)**.
Additive basic rotat. (W-CS)	■ **Status** ■ **(°)** The **Additive basic rotat. (W-CS)** function is in effect in the workpiece coordinate system **W-CS**. Entries are in degrees. **Further information:** "Workpiece coordinate system W-CS", Page 940
Shift (W-CS)	■ **Status** ■ **X** **Shift (W-CS)** in the X axis The **Shift (W-CS)** function is also available for the other linear axes **Y** and **Z**.
Mirroring (W-CS)	■ **Status** ■ **X** **Mirroring (W-CS)** in the X axis The **Mirroring (W-CS)** function is also available for the other linear axes **Y** and **Z**, as well as for the rotary axes available in the respective machine kinematics.
Rotation (WPL-CS)	■ **Status** ■ **(°)** **Rotation (WPL-CS)** in degrees The **Rotation (WPL-CS)** function is in effect in the working plane coordinate system **WPL-CS**. Entries are in degrees. **Further information:** "Working plane coordinate system WPL-CS", Page 942
Shift (mW-CS)	■ **Status** ■ **X** **Shift (mW-CS)** in the X axis The **Shift (mW-CS)** function is also available for the other linear axes **Y** and **Z**, as well as for the rotary axes available in the respective machine kinematics.

Area	Contents
Handwheel superimp.	■ **Status** ■ **Coordinate system** This area contains the selected coordinate system for **Handwheel superimp.**, such as the machine coordinate system **M-CS**. ■ **X** ■ **Y** ■ **Z** ■ **A (°)** ■ **B (°)** ■ **C (°)** ■ **VT**
Feed rate factor	If the **Feed rate factor** function is active, the control displays the defined percentage in this field. If the **Feed rate factor** function is not active, the control displays **100.00 %** in this field.

LBL tab

On the **LBL** tab the control shows information about program section repeats and subprograms.

Further information: "Subprograms and program section repeats with the label LBL", Page 332

Area	Contents
Subprograms	■ **Blk. no.** Block number of the call ■ **LBL no./Name** Called label
Repetitions	■ **Blk. no.** ■ **LBL no./Name** ■ **Program-section repeat** Number of repetitions still to be performed, e.g. 4/5

M tab

On the **M** tab the control shows information about active miscellaneous functions.

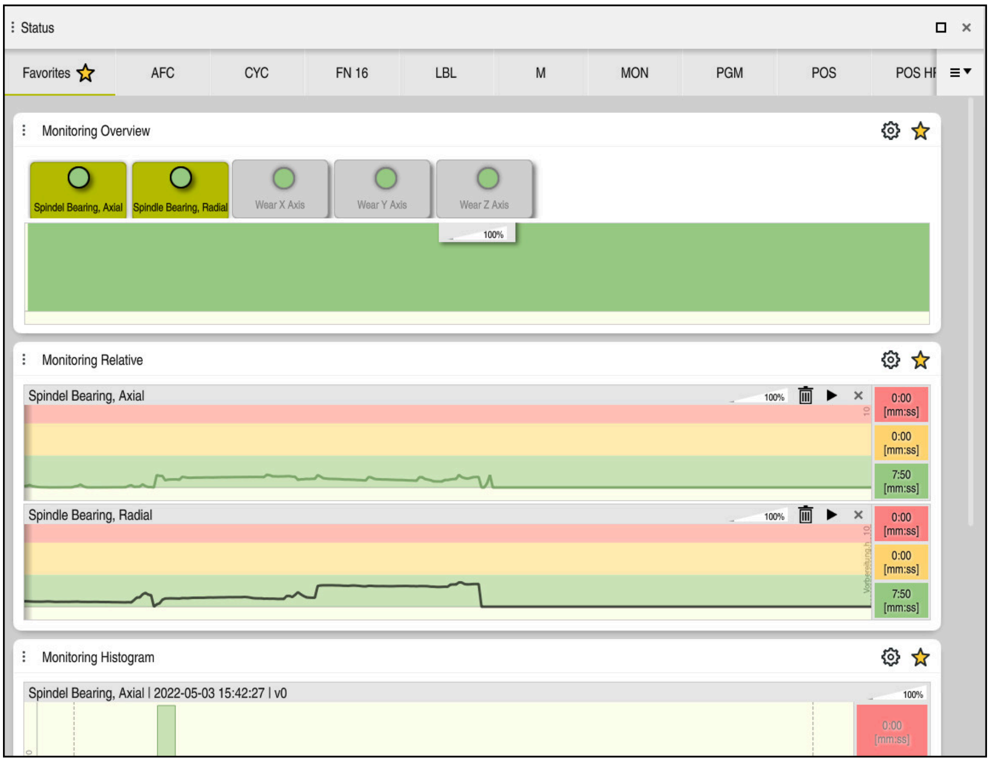
Further information: "Miscellaneous Functions", Page 1223

Area	Contents
Active M functions	■ **Function** Active miscellaneous functions, such as **M3** ■ **Description** Descriptive text about the respective miscellaneous function. Refer to your machine manual. Only the machine manufacturer can create a descriptive text for machine-specific miscellaneous functions.

MON tab (option 155)

On the **MON** tab the control shows information about machine components defined to be monitored through Component Monitoring (option 155).

> Refer to your machine manual.
> The machine manufacturer specifies which machine components are monitored, and to what extent.



MON tab with configured spindle speed monitoring

Area	Contents
Monitoring Overview	The control displays the machine components defined for monitoring. By selecting a component, you hide or show whether it is being monitored.
Monitoring Relative	The control displays the monitoring information for the components being shown in the **Monitoring Overview** area. ■ Green: component works under conditions defined as safe ■ Yellow: component works under warning zone conditions ■ Red: component is overloaded In the **Display settings** window you can select which component the control shows.
Monitoring Histogram	The control shows a graphical evaluation of previous monitoring sessions.

Use the **Settings** symbol to open the **Display settings** window. You can define the height of each graphical depiction for each area.

PGM tab

On the **PGM** tab the control shows information about the program run.

Area	Contents
Parts counter	■ **Quantity** Actual value and nominal value of the parts counter defined with the **FUNCTION COUNT** function **Further information:** "Defining counters with FUNCTION COUNT", Page 1308
Program run time	■ **Runtime** Run time of the NC program in hh:mm:ss ■ **Dwell time** Countdown of the waiting time in seconds from the following functions: ■ Cycle **9 DWELL TIME** ■ Parameter **Q210 DWELL TIME AT TOP** ■ Parameter **Q211 DWELL TIME AT DEPTH** ■ Parameter **Q255 DWELL TIME** **Further information:** "Display of the program run time", Page 166
Programs called	Path of the main program as well as called NC programs including the path
Pole/circle center	Programmed axes and values of the circle center point **CC**

POS tab

On the **POS** tab the control shows information about positions and coordinates.

Area	Contents
Position display, e.g., **Actual reference position (RFACTL)**	In this area the control shows the current position of all axes that are present. You can choose between the following views in the position display: ■ **Nominal pos. (NOML)** ■ **Actual pos. (ACT)** ■ **Nominal reference position (RFNOML)** ■ **Actual reference position (RFACTL)** ■ **Servo lag (LAG)** ■ **Handwheel superimposed (M118)** **Further information:** "Position displays", Page 167
Feed and Speed	■ Active **Feed** in mm/min ■ Active **Feed-rate override** in % ■ Active **Rapid-traverse override** in % ■ Active **Spindle speed** in rpm ■ Active **Spindle override** in % ■ Active **Miscellaneous function** in reference to the spindle, such as **M3**
Orientation of the working plane	Spatial angles or axis angles for the active working plane **Further information:** "Tilting the working plane with PLANE functions (option 8)", Page 979 If axis angles are active, the control displays in this area only the values of the physically present axes. The defined values are in the window **3-D rotation** **Further information:** "3D ROT setting", Page 1024
OEM transformation	The machine manufacturer can define an OEM transformation for special turning kinematics. **Further information:** "Definitions", Page 164
Basic transformations	In this area the control shows the values of the active workpiece preset and active transformations in linear and rotary axes, such as a transformation in the X axis with the function **TRANS DATUM**. **Further information:** "Preset management", Page 949
Special turning transformations	Transformations relevant for turning operations (option 50), such as a defined **precession angle** from the following sources: ■ Defined by the machine manufacturer ■ Cycle **800 ADJUST XZ SYSTEM** ■ Cycle **801 RESET ROTARY COORDINATE SYSTEM** ■ Cycle **880 GEAR HOBBING**
Active traverse ranges	Active traverse range, such as Limit 1 for traverse range 1 Traverse ranges are machine-specific. If no traverse range is active, then **Traverse range not defined** is displayed in this area.

Area	Contents
Active kinemat.	Name of the active machine kinematics

POS HR tab

On the **POS HR** tab the control shows information about handwheel superimpositioning.

Area	Contents
Coordinate system	■ **Machine (M-CS)** With **M118** the handwheel superimpositioning is always in effect in the machine coordinate system **M-CS**. **Further information:** "Activating handwheel superimpositioning with M118", Page 1239 With the Global Program Settings (GPS, option 44) the coordinate system can be chosen. **Further information:** "Global Program Settings (GPS, option 44)", Page 1132
Handwheel superimp.	■ **Max. val.** Maximum value of the individual axes programmed in **M118** or in the **GS** workspace ■ **Actl.val.** Current superimpositioning

QPARA tab

On the **QPARA** tab the control shows information about the defined variables.

Further information: "Variables: Q, QL, QR and QS parameters", Page 1268

You use the **Parameter list** window to define which variables the control shows in the areas.

Further information: "Defining the contents of the QPARA tab", Page 169

Area	Contents
Q parameter	Shows the values of the selected Q parameters
QL parameter	Shows the values of the selected QL parameters
QR parameter	Shows the values of the selected QR parameters
QS parameter	Shows the contents of the selected QS parameters

Tables tab

On the **Tables** tab the control shows information about the active tables for program run or the simulation.

Area	Contents
Active tables	In this area the control shows the path for the following active tables: ■ Tool table ■ Turning tool table ■ Preset table ■ Datum table ■ Pocket table ■ Touch probe table ■ Grinding tool table ■ Dressing tool table

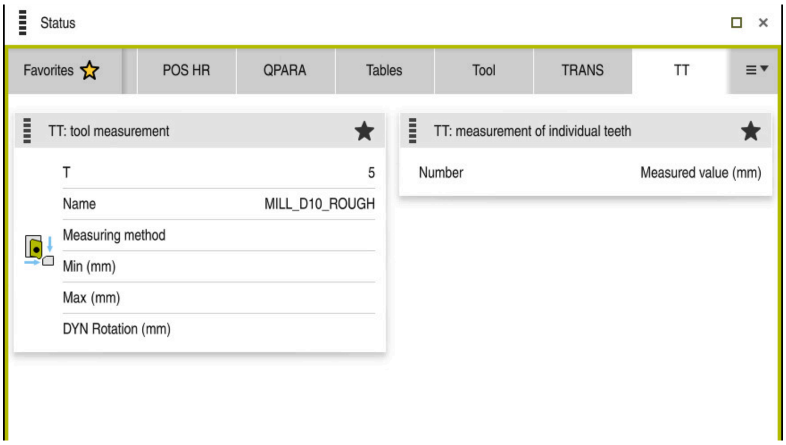
TRANS tab

On the **TRANS** tab the control shows information about active transformations in the NC program.

Area	Contents
Active datum	■ Path of the selected datum table ■ Line number of the selected datum table ■ **Doc** Contents of the **DOC** column of the datum table
Active datum shift	Datum shift that was defined with the **TRANS DATUM** function **Further information:** "Datum shift with TRANS DATUM", Page 972
Mirrored axes	Axes mirrored with either the **TRANS MIRROR** function or Cycle **8 MIRRORING** **Further information:** "Mirroring with TRANS MIRROR", Page 973 **Further information:** "Cycle 8 MIRRORING", Page 959
Active angle of rotation	Rotation angle defined with either the **TRANS ROTATION** function or Cycle **10 ROTATION** **Further information:** "Rotations with TRANS ROTATION", Page 975 **Further information:** "Cycle 10 ROTATION ", Page 961
Orientation of the working plane	Spatial angles for the active working plane **Further information:** "Tilting the working plane with PLANE functions (option 8)", Page 979
Center coordinates	Center of scaling that was defined with Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** "Cycle 26 AXIS-SPECIFIC SCALING ", Page 963
Active scaling factors	Scaling factors that were defined for the individual axes with the **TRANS SCALE** function, Cycle **11 SCALING FACTOR** or Cycle **26 AXIS-SPECIFIC SCALING** **Further information:** "Scaling with TRANS SCALE", Page 977 **Further information:** "Cycle 11 SCALING FACTOR ", Page 962 **Further information:** "Cycle 26 AXIS-SPECIFIC SCALING ", Page 963

TT tab

On the **TT** tab the control shows information about measurements performed with a TT tool touch probe.



TT tab with values from the measurement of tool cutting edges of a milling cutter

<table>
<tr><th>Area</th><th>Contents</th></tr>
<tr><td>TT Tool to be measured</td><td>

T
Tool number
Name
Tool name
Measure what?
Measurement method chosen for tool measurement, such as Length
Min (mm)
When measuring milling cutters, in this area the control shows the smallest measured value of a cutting edge.
When measuring turning tools (option 50), in this area the control shows the smallest measured tipping angle. The value of the angle can also be negative.
Further information: "Definitions", Page 164
Max (mm)
When measuring milling cutters, in this area the control shows the greatest measured value of a cutting edge.
When measuring turning tools, in this area the control shows the greatest measured tipping angle. The value of the angle can also be negative.
DYN Rotation (mm)
When measuring milling cutters with a rotating spindle, the control shows values in this area.
When measuring turning tools, the value DYN ROTATION describes the tolerance for the tipping angle. If the tolerance for the tipping angle is exceeded during calibration, the control marks the affected value in the MIN or MAX fields with an *.

In the optional machine parameter tippingTolerance (no. 114206) you define the tipping angle tolerance. The control will determine the tipping angle automatically only if a tolerance is defined.
</td></tr>
<tr><td>TT Tool cutting edges</td><td>Number
List of the measurements performed and the measured values of the individual cutting edges</td></tr>
</table>

Tool tab

On the **Tool** tab the control shows information about the active tool, depending on the tool type.

Further information: "Tool types", Page 255

Contents for dressing, milling, and grinding tools (option 156)

Area	Contents
Tool information	■ **T** Tool number ■ **Name** Tool name ■ **Doc** Note on the tool
Tool geometry	■ **L** Tool length ■ **R** Tool radius ■ **R2** Corner radius of the tool
Tool allowances	■ **DL** Delta value for the tool length ■ **DR** Delta value for the tool radius ■ **DR2** Delta value for the corner radius of the tool
Tool ages	■ **Cur. time (h:m)** Time in hours and minutes the tool has been engaged ■ **Time 1 (h:m)** Service life of the tool ■ **Time 2 (h:m)** Maximum service life at tool call
Replacement tool	■ **RT** Tool number of the replacement tool ■ **Name** Tool name of the replacement tool
Tool type	■ **Tool Axis** Tool axis programmed in the tool call (e.g., **Z**) ■ **Type** Tool type of the active tool (e.g., **DRILL**)

Other contents for turning tools (option 50)

Area	Contents
Tool geometry	■ **ZL (mm)** Tool length in Z direction ■ **XL (mm)** Tool length in X direction ■ **RS (mm)** Cutter radius ■ **YL (mm)** Tool length in Y direction
Tool allowances	■ **DZL (mm)** Delta value in Z direction ■ **DXL (mm)** Delta value in X direction ■ **DRS (mm)** Delta value for the cutter radius ■ **DCW (mm)** Delta value for the width of the recessing tool
Tool type	■ **Tool Axis** ■ **TO** Tool orientation ■ **Type** Tool type (e.g. **TURN**)

Definitions

OEM transformations for special turning kinematics
Machine manufacturers can define OEM transformations for special turning kinematics. Machine manufacturers need these transformations for milling-turning machines that have a different orientation than the tool coordinate system in the home position of their axes.

Tipping angle
If a TT tool touch probe with a cuboid contact cannot be clamped to a machine table so that it is level, the angular offset must be compensated for. This offset is the tipping angle.

Angle of misalignment
In order to exactly measure with TT tool touch probes with a cuboid contact, the misalignment on the machine table relative to the main axis must be compensated for. This offset is the angle of misalignment.

4.5 Simulation status workspace

Application

You can call additional status displays in the **Editor** operating mode in the **Simulation status** workspace. In the **Simulation status** workspace the control shows data based on the simulation of the NC program.

Description of function

The following tabs are available in the **Simulation status** workspace:

- **Favorites**
  Further information: "Favorites tab", Page 149
- **CYC**
  Further information: "CYC tab", Page 151
- **FN16**
  Further information: "FN16 tab", Page 151
- **LBL**
  Further information: "LBL tab", Page 153
- **M**
  Further information: "M tab", Page 154
- **PGM**
  Further information: "PGM tab", Page 156
- **POS**
  Further information: "POS tab", Page 157
- **QPARA**
  Further information: "QPARA tab", Page 158
- **Tables**
  Further information: "Tables tab", Page 159
- **TRANS**
  Further information: "TRANS tab", Page 160
- **TT**
  Further information: "TT tab", Page 161
- **Tool**
  Further information: "Tool tab", Page 163

4.6 Display of the program run time

Application

The control calculates the duration of all traverse movements and displays them together as the **Program run time**. In the **Status** workspace the control takes traverse movements and dwell times into account. The control does not take dwell times into account in the **Simulation status** workspace.

Description of function

The control displays the program run time in the following areas:

- **PGM** tab of the **Status** workspace
- Status overview on the control bar
- **PGM** tab of the **Simulation status** workspace
- **Simulation** workspace in the **Editor** operating mode

Modify the **Settings** in the **Program run time** area in order to influence the calculated program run time.

Further information: "PGM tab", Page 156

The control opens a selection menu with the following functions:

Function	Meaning
Save	Save the current value under **Runtime**
Addition	Add the saved time to the value under **Runtime**
Resetting	Reset the saved time and the contents of the **Program run time** area to zero

The control counts the time during which the **Control-in-operation** symbol is green. The control adds the time from the **Program Run** operating mode and the **MDI** application.

The following functions reset the program run time:

- Selecting a new NC program for program run
- The **Reset program** button
- The **Resetting** function in the **Program run time** area

Notes

- In the machine parameter **operatingTimeReset** (no. 200801) the machine manufacturer defines whether the control resets the program run time when the program is started.
- The control cannot simulate the run time of machine-specific functions such as tool changing. That is why this function is only partially suitable for calculating the production time in the **Simulation** workspace.
- In the **Program Run** operating mode, the control displays the exact time of the NC program while taking all machine-specific actions into account.

Definition

Control-in-operation:
The control uses the **Control-in-operation** symbol to show the machining status of the NC program or NC block:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

When the control bar is expanded, the control shows additional information about the current status, such as **Active, feed rate at zero**.

4.7 Position displays

Application

The control offers various modes in the position display, for example values from different reference systems. You can choose one of the modes available based on the application.

Description of function

The control has position displays in the following areas:

- **Positions** workspace
- Status overview on the control bar
- **POS** tab of the **Status** workspace
- **POS** tab of the **Simulation status** workspace

On the **POS** tab of the **Simulation status** workspace the control always shows the **Nominal pos. (NOML)** mode. In the **Status** and **Positions** workspaces you can choose the mode of the position display.

The control offers the following modes for the position display:

Mode	Meaning
Nominal pos. (NOML)	This mode shows the value of the currently calculated target position in the input coordinate system **I-CS**. When the machine moves the axes, the control compares the coordinates of the measured actual position with the calculated nominal position in predefined time intervals. The nominal position is the position at which the axes should be located at the time of comparison, based on the calculation. The **Nominal pos. (NOML)** and **Actual pos. (ACT)** modes differ solely with regard to the servo lag.
Actual pos. (ACT)	This mode shows the currently measured tool position in the input coordinate system **I-CS**. The actual position is the measured position of the axes, as determined by encoders at the time of comparison.

Mode	Meaning
Nominal reference position (RFNOML)	This mode shows the calculated target position in the machine coordinate system **M-CS**. The **Nominal reference position (RFNOML)** and **Actual reference position (RFACTL)** modes differ solely with regard to the servo lag.
Actual reference position (RFACTL)	This mode shows the currently measured tool position in the machine coordinate system **M-CS**.
Servo lag (LAG)	This mode shows the difference between the calculated nominal position and the measured actual position. The control determines the difference in predefined time intervals.
Handwheel superimposed (M118)	This mode shows the values that you move using the **M118** miscellaneous function. **Further information:** "Activating handwheel superimpositioning with M118", Page 1239

Refer to your machine manual.

In the machine parameter **progToolCallDL** (no. 124501) the machine manufacturer defines whether the position display takes the delta value **DL** from the tool call into account. The modes **NOML.** and **ACTL.** as well as **RFNOML** and **RFACTL** then differ from each other by the value **DL**.

4.7.1 Switching the position display mode

To switch the position display mode in the **Status** workspace:

- Select the **POS** tab
- Select **Settings** in the position display area
- Select the desired mode for the position display, for example **Actual pos. (ACT)**
- > The control displays the positions in the selected mode.

Notes

- The machine parameter **CfgPosDisplayPace** (no. 101000) defines the display accuracy by the number of decimal places.
- When the machine moves the axes, the control displays the distances-to-go of the individual axes with a symbol and the appropriate value next to the current position.

  Further information: "Axis display and position display", Page 142

4.8 Defining the contents of the QPARA tab

On the **QPARA** tab of the **Status** and **Simulation status** workspaces you can define which variables the control shows.

Further information: "QPARA tab", Page 158

To define the contents of the **QPARA** tab, proceed as follows:

- Select the **QPARA** tab
- Select the **Settings** in the desired area, such as QL parameters
- > The control opens the **Parameter list** window.
- Enter numbers, such as **1,3,200-208**
- OK Press **OK**
- > The control displays the values of the defined variables.

- Use commas to separate single variables and connect sequential variables with a hyphen.
- The control always shows eight decimal places on the **QPARA** tab. For example, the control shows the result of **Q1 = COS 89.999** as 0.00001745. Very large and very small values are shown in exponential notation. The control shows the result of **Q1 = COS 89.999 * 0.001** as +1.74532925e-08, with e-08 corresponding to the factor of 10^{-8}.
- For variable texts in QS parameters the control shows the first 30 characters, i.e. the contents might be truncated.

5

Powering On and Off

5.1 Powering on

Application

After using the main switch to power on the machine, the control's boot process begins. The following steps may differ depending on the machine; for example, whether absolute or incremental position encoders are used.

Refer to your machine manual.
Switching on the machine and traversing the reference points can vary depending on the machine tool.

Related topics

- Absolute and incremental position encoders
  Further information: "Position encoders and reference marks", Page 187

Description of function

DANGER

Caution: hazard to the user!

Machines and machine components always pose mechanical hazards. Electric, magnetic, or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

- Read and follow the machine manual
- Read and follow the safety precautions and safety symbols
- Use the safety devices

Power-on of the control begins with the power supply.

After booting, the controls checks the machine status, e.g.:

- Positions identical to before switching off the machine
- Safety features are ready, such as the emergency stop
- Functional safety

If the control registers an error during or after booting, it issues an error message.

The following step differs depending on position encoders on the machine:

- Absolute position encoders
  If the machine has absolute position encoders, the control is in the **Start/Login** application after power-on.
- Incremental position encoders
  If the machine has incremental position encoders, you must traverse the reference points in the **Move to ref. point** application. Once all axes have been referenced, the control is in the **Manual operation** application.
  Further information: "Referencing workspace", Page 174
  Further information: "Manual operation application", Page 180

5.1.1 Powering the machine and the control on

To switch the machine on:

- ▶ Switch the power supply of the control and of the machine on
- \> The control is in start-up mode and shows the progress in the **Start/Login** workspace.
- \> The control displays the **Power interrupted** dialog in the **Start/Login** workspace.

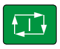
OK

- ▶ Press **OK**
- \> The control compiles the PLC program.

I

- ▶ Switch the machine control voltage on
- \> The control checks the functioning of the emergency stop circuit.
- \> If the machine is equipped with absolute linear and angle encoders, the control is now ready for operation.
- \> If the machine is equipped with incremental linear and angle encoders, the control opens the **Move to ref. point** application.

  Further information: "Referencing workspace", Page 174



- ▶ Press the **NC Start** key
- \> The control moves to all necessary reference points.
- \> The control is ready for operation and the **Manual operation** application is open.

  Further information: "Manual operation application", Page 180

Notes

NOTICE

Danger of collision!

When the machine is switched on, the control tries to restore the switch-off status of the tilted plane. This is prevented under certain conditions. For example, this applies if axis angles are used for tilting while the machine is configured with spatial angles, or if you have changed the kinematics.

- If possible, reset tilting before shutting the system down
- Check the tilted condition when switching the machine back on

NOTICE

Danger of collision!

Failure to notice deviations between the actual axis positions and those expected by the control (saved at shutdown) can lead to undesirable and unexpected axis movements. There is risk of collision during the reference run of further axes and all subsequent movements!

- Check the axis positions
- Only confirm the pop-up window with **YES** if the axis positions match
- Despite confirmation, at first only move the axis carefully
- If there are discrepancies or you have any doubts, contact your machine tool builder

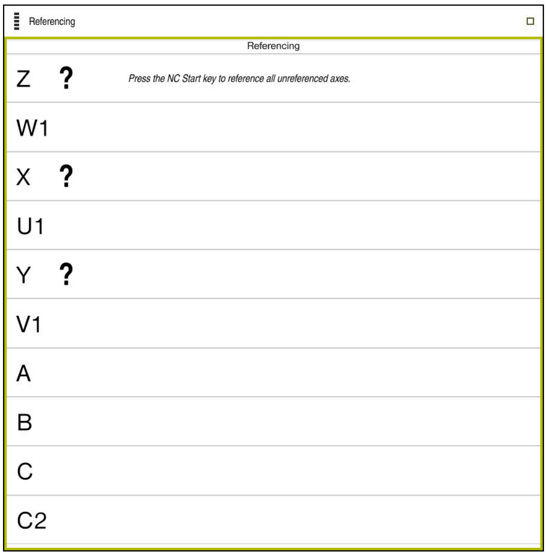
5.2 Referencing workspace

Application

On machines with incremental linear and angle encoders, the control shows in the **Referencing** workspace which axes need to be referenced.

Description of function

The **Referencing** workspace is always open in the **Move to ref. point** application. If reference points are to be traversed when powering-on the machine, then the control opens this application automatically.



Referencing workspace with axes to be referenced

The control displays a question mark behind all axes that need to be referenced.

Once all axes have been referenced, the control closes the **Move to ref. point** application and switches to the **Manual operation** application.

5.2.1 Axis reference run

To reference the axes in the prescribed sequence:

- ▶ Press the **NC start** key
- > The control moves to the reference points.
- > The control switches to the **Manual operation** application.

To reference the axes in any sequence:

X+

- ▶ Press and hold the axis direction button for each axis until the reference point has been traversed
- > The control switches to the **Manual operation** application.

Notes

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision when referencing the axes.

- Pay attention to the information on the screen
- If necessary, move to a safe position before referencing the axes
- Watch out for possible collisions

- You cannot switch to the **Program Run** operating mode as long as reference points still need to be traversed.
- If you intend only to edit or simulate NC programs, you can switch to the **Editor** operating mode without referencing the axes. You can still traverse the reference points at a later time.

Notes about traversing reference points in a tilted working plane

If the **Tilt working plane** (option 8) function was active before the control was shut down, then the control will automatically reactivate this function after restarting. This means that movements via the axis keys take place in the tilted working plane.

Before traversing the reference points, you must deactivate the **Tilt the working plane** function; otherwise, the control will interrupt the process with a warning. You can also home axes not activated in the current kinematic model without needing to deactivate **Tilt the working plane**, such as a tool magazine.

Further information: "3-D rotation window (option 8)", Page 1022

5.3 Powering off

Application

To avoid losing data, shut down the control before powering-off the machine.

Description of function

You shut down the control in the **Start/Login** application of the **Home** operating mode.

If you select the **Shut down** button, the control opens the **Shut down** window. You choose whether to shut down the control or restart it.

5.3.1 Shutting down the control and powering-off the machine

To power-off the machine:

- Select the **Home** operating mode

Shut down

- Select **Shut down**
- > The control opens the **Shut down** window.

Shut down

- Select **Shut down**
- > The control shuts down.
- > Once shutdown has concluded, the control displays the text **Now you can switch off.**
- Switch off the main switch of the machine

Notes

NOTICE

Caution: Data may be lost!

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss no matter what state the control was in!

- Always shut down the control
- Only operate the main switch after being prompted on the screen

- Different machines have different power-off procedures.
  Refer to your machine manual.
- Applications on the control can delay the shutdown, such as a connection with **Remote Desktop Manager** (option 133)
  "Remote Desktop Manager window (option 133)"

6

Manual Operation

6.1 Manual operation application

Application

In the **Manual operation** application you can manually move the axes and set up the machine.

Related topics

- Moving the machine axes
  Further information: "Moving the machine axes", Page 182
- Incremental jog positioning of machine axes
  Further information: "Incremental jog positioning of axes", Page 183

Description of function

The **Manual operation** application offers the following workspaces:

- **Positions**
- **Simulation**
- **Status**

The function bar in the **Manual operation** application contains the following buttons:

Button	Meaning
Handwheel	The control displays this switch if a handwheel is configured for the control. If the handwheel is active, the operating mode's icon in the sidebar changes. **Further information:** "Electronic Handwheel", Page 1881
M	Define a miscellaneous function **M** or use the selection window to choose one and activate it with the **NC start** key. **Further information:** "Miscellaneous Functions", Page 1223
S	Define the spindle speed **S** activate it with the **NC start** key, and also switch on the spindle. **Further information:** "Spindle speed S", Page 281
F	Define the feed rate **F** and activate it with the **OK** button. **Further information:** "Feed rate F", Page 282
T	Define a tool **T** or use the selection window to choose one and insert it with the **NC start** key. **Further information:** "Tool call", Page 277
Jog increment	Define the jog increment **Further information:** "Incremental jog positioning of axes", Page 183
Set the preset	Enter and set a preset **Further information:** "Preset management", Page 949
3D ROT	The control opens a window for the 3D rotation settings (option 8). "3-D rotation window (option 8)"
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** "Q parameter list window", Page 1271
DCM	The control opens the **Collision monitoring (DCM)** window, where you can activate or deactivate Dynamic Collision Monitoring (DCM, option 40). **Further information:** "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 1088

Note

The machine manufacturer defines which miscellaneous functions are available on the control and which are allowed in the **Manual operation** application.

6.2 Moving the machine axes

Application

You can use the control to move the machine axes manually, such as pre-positioning for a manual touch probe function.

Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1425

Related topics

- Programming traverse movements
  Further information: "Path Functions", Page 289
- Executing traverse movements in the **MDI** application
  Further information: "Application MDI", Page 1759

Description of function

The control offers the following methods for moving axes manually:

- Axis-direction keys
- Incremental jog positioning with the **Jog increment** button
- Traversing with electronic handwheels
  Further information: "Electronic Handwheel", Page 1881

The control displays the current contouring feed rate in the status display while the machine axes are in motion.

Further information: "Status Displays", Page 139

You can change the contouring feed rate with the **F** button in the **Manual operation** application and with the feed-rate potentiometer.

A traverse job is active on the control as soon as an axis moves. The control shows the status of the traverse job with the **Control-in-operation** icon in the status overview.

Further information: "Status overview on the control bar", Page 147

6.2.1 Using axis keys to move the axes

To move an axis manually with the axis keys:

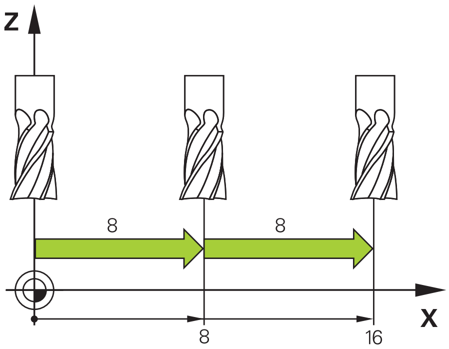
- ▶ Select an operating mode, e.g. **Manual**
- ▶ Select an application, e.g, **Manual operation**

X+
- ▶ Press the axis key of the desired axis
- > The control moves the axis as long as you press the key.

i	If you hold the axis key pressed down and simultaneously press the **NC start** key, the control moves the axis at a continuous feed rate. You have to end traverse movement with the **NC stop** key. You can move more than one axis at a time.

6.2.2 Incremental jog positioning of axes

With incremental jog positioning you can move a machine axis by a preset distance. The input range for the infeed is from 0.001 mm to 10 mm.



To position an axis incrementally:

- Select the **Manual** operating mode
- Select the **Manual operation** application

Jog increment

- Select **Jog increment**
- > The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.
- Enter the jog increment for linear axes and rotary axes

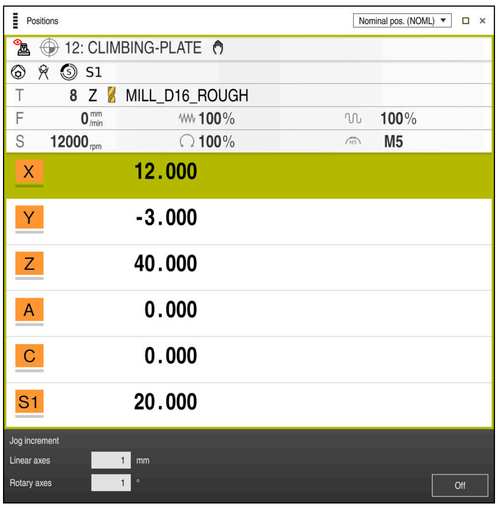
X+

- Press the axis key of the desired axis
- > The control positions the axis in the selected direction by the defined jog increment.

Jog increment On

- Select **Jog increment On**
- > The control ends incremental jog positioning and closes the **Jog increment** area in the **Positions** workspace.

> You can also end incremental jog positioning with the **Off** button in the **Jog increment** area.



Positions workspace with the **Jog increment** area active

Note

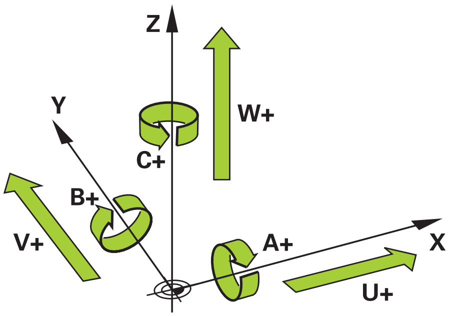
When moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.

7

NC and Programming Fundamentals

7.1 NC fundamentals

7.1.1 Programmable axes



The programmable axes of the control are in accordance with the axis definitions specified in DIN 66217.

The programmable axes are designated as follows:

Main axis	Parallel axis	Rotary axis
X	**U**	**A**
Y	**V**	**B**
Z	**W**	**C**

> Refer to your machine manual.
> The number, designation and assignment of the programmable axes depend on the machine.
> Your machine manufacturer can define further axes, such as PLC axes.

7.1.2 Designation of the axes on milling machines

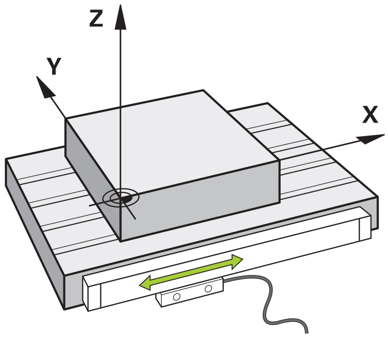
The axes **X**, **Y** and **Z** on your milling machine are designated as the main axis (1st axis), secondary axis (2nd axis) and tool axis. The main axis and secondary axis define the working plane.

The axes are associated as follows:

Main axis	Secondary axis	Tool axis	Working plane
X	**Y**	**Z**	**XY**, also **UV**, **XV**, **UY**
Y	**Z**	**X**	**YZ**, also **WU**, **ZU**, **WX**
Z	**X**	**Y**	**ZX**, also **VW**, **YW**, **VZ**

7.1.3 Position encoders and reference marks

Fundamentals



The position of the machine axes is ascertained with position encoders. As a rule, linear axes are equipped with linear encoders. Rotary tables and rotary axes feature angle encoders.

The position encoders detect the positions of the tool or machine table by generating an electrical signal during movement of an axis. The control ascertains the position of the axis in the current reference system from this electrical signal.

Further information: "Reference systems", Page 934

Position encoders can measure these positions through different methods:

- Absolutely
- Incrementally

The control cannot determine the position of the axes while the power is interrupted. Absolute and incremental position encoders behave differently once power is restored.

Absolute position encoders

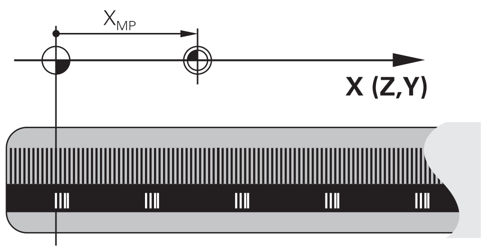
On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

Incremental position encoders

Incremental position encoders need to find the distance between the current position and a reference mark in order to determine the actual position. Reference marks indicate a machine-based reference point. A reference mark must be traversed in order to determine the current position after a power interruption.

If the position encoders feature distance-coded reference marks, then you need to move the linear encoders of the axes by no more than 20 mm. On angle encoders this distance is no more than 20 °.

Further information: "Axis reference run", Page 175

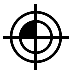
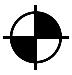
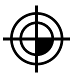
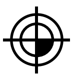


7.1.4 Presets in the machine

The following table contains an overview of the presets in the machine or on the workpiece.

Related topics

- Presets on the tool
  Further information: "Presets on the tool", Page 244

Icon	Preset
	**Machine datum** The machine datum is a fixed point defined in the machine configuration by the machine manufacturer. The machine datum is the origin of the machine coordinate system **M-CS**. **Further information:** "Machine coordinate system M-CS", Page 935 If you program **M91** in an NC block, the defined values are referenced to the machine datum. **Further information:** "Traversing in the machine coordinate system M-CS with M91", Page 1228
 M92-ZP	**M92 datum M92-ZP (zero point)** The **M92** datum is a fixed point defined relative to the machine datum by the machine manufacturer in the machine configuration. The **M92** datum is the origin of the **M92** coordinate system. If you program **M92** in an NC block, the defined values are referenced to the **M92** datum. **Further information:** "Traversing in the M92 coordinate system with M92", Page 1229
	Tool change position The tool change position is a fixed point defined relative to the machine datum by the machine manufacturer in the tool-change macro.
	**Reference point** The reference point is a fixed point for initializing position encoders. **Further information:** "Position encoders and reference marks", Page 187 If the machine has incremental position encoders, the axes must traverse the reference point after booting. **Further information:** "Axis reference run", Page 175
	**Workpiece preset** With the workpiece preset you define the origin of the workpiece coordinate system **W-CS**. **Further information:** "Workpiece coordinate system W-CS", Page 940 The workpiece preset is defined in the active row of the preset table. You determine the workpiece preset with a 3D touch probe, for example. **Further information:** "Preset management", Page 949 If no transformations are defined, the entries in the NC program refer to the workpiece preset.
	**Workpiece datum** You define the workpiece datum with transformations in the NC program, for example with **TRANS DATUM** or a datum table. The entries in the NC program refer to the workpiece datum. If no transformations are defined in the NC program, the workpiece datum corresponds to the workpiece preset. If you tilt the working plane (option 8), the workpiece datum is the point around which the workpiece is rotated.

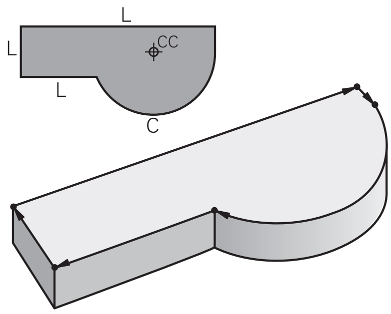
7.2 Programming possibilities

7.2.1 Path functions

Use the path functions to program contours.

A workpiece contour consists of several contour elements, such as straight lines and circular arcs. You use path functions, such as straight line **L**, to program tool movements for these contours.

Further information: "Fundamentals of path functions", Page 293



7.2.2 Graphical programming

As an alternative to Klartext programming you can program contours graphically in the **Contour graphics** workspace.

You can create 2D sketches by drawing lines and arcs and then export the contour to an NC program.

You can import existing contours from an NC program for graphical editing.

Further information: "Graphical Programming", Page 1337

7.2.3 Miscellaneous functions M

You can use miscellaneous functions to control the following actions:

- Program run, e.g. **M0** Program STOP
- Machine functions, e.g. **M3** Spindle ON clockwise
- Contouring behavior of the tool, e.g. **M197** Corner rounding

Further information: "Miscellaneous Functions", Page 1223

7.2.4 Subprograms and program-section repeats

Subprograms and program-section repeats enable you to program a machining sequence once and then run it as often as necessary.

Program sections that are defined in a label can be directly executed repeatedly as program-section repeats, or can be called as a subprogram at defined locations in the main program.

If you wish to execute a specific NC program section only under certain conditions, you also define this machining sequence as a subprogram.

Within an NC program you can call a separate NC program for execution.

Further information: "Subprograms and program section repeats with the label LBL", Page 332

7.2.5 Programming with variables

In an NC program, variables are used as placeholders for numerical values or texts. A numerical value or text is assigned to a variable elsewhere.

In the **Q parameter list** window, you can see and edit the numerical values and texts of the individual variables.

Further information: "Q parameter list window", Page 1271

You can use the variables to program mathematical functions that control program execution or describe a contour.

You can also use variable programming, for example, to save and process measurement results determined by the 3D touch probe during program execution.

Further information: "Variables: Q, QL, QR and QS parameters", Page 1268

7.2.6 CAM programs

You can also optimize and execute externally created NC programs on the control.

You use CAD (**Computer-Aided Design**) to create geometric models of the workpieces to be produced.

In a CAM system (**Computer-Aided Manufacturing**) you then define how the CAD model will be produced. You can use an internal simulation to check the resulting tool paths, which are not control-specific.

With a postprocessor in the CAM system you then generate the control- and machine-specific NC programs. This results not only in programmable path functions, but also in splines (**SPL**) and straight lines **LN** with surface normal vectors.

Further information: "Multiple-Axis Machining", Page 1177

7.3 Programming fundamentals

7.3.1 Contents of an NC program

Application

You use NC programs to define the movements and behavior of your machine. NC programs consist of NC blocks that contain the syntax elements of the NC functions. With the HEIDENHAIN Klartext programming language, the control supports you by showing a dialog with information about the required content for every syntax element.

Related topics

- Creating a new NC program

  Further information: "Creating a new NC program", Page 112
- NC programs using CAD files

  Further information: "CAM-generated NC programs", Page 1208
- Structure of an NC program for contour machining

  Further information: "Structure of an NC program", Page 115

Description of function

You create NC programs in the **Editor** operating mode in the **Program** workspace.

Further information: "Program workspace", Page 194

The first and last NC blocks of the NC program contain the following information:

- Syntax **BEGIN PGM** or **END PGM**
- Name of the NC program
- Unit of measure of the NC program (mm or inches)

The control automatically inserts the **BEGIN PGM** and **END PGM** NC blocks when creating the NC program. You cannot delete these NC blocks.

The NC blocks created after **BEGIN PGM** contain the following information:

- Workpiece blank definition
- Tool calls
- Approaching a safe position
- Feed rates and spindle speeds
- Traverse movements, cycles and other NC functions

0 BEGIN PGM EXAMPLE MM	; Start of program
1 BLK FORM 0.1 Z X-50 Y-50 Z-20	
2 BLK FORM 0.2 X+50 Y+50 Z+0	; NC function for workpiece blank definition, consisting of two NC blocks
3 TOOL CALL 5 Z S3200 F300	; NC function for tool call
4 L Z+100 R0 FMAX M3	; NC function for straight-line traverse
*** - ...**	
11 M30	; NC function for concluding the NC program
12 END PGM EXAMPLE MM	; End of program

Syntax component	Meaning
NC block	**4 TOOL CALL 5 Z S3200 F300** An NC block consists of the block number and the syntax of the NC function. An NC block can consist of multiple lines, such as with cycles. The control numbers the NC blocks in ascending sequence.
NC function	**TOOL CALL 5 Z S3200 F300** You use NC functions to define the behavior of the control. The block number is not a part of the NC functions.
Syntax initiator	**TOOL CALL** The syntax initiator clearly designates each NC function. Syntax initiators are used in the **Insert NC function** window. **Further information:** "Inserting NC functions", Page 203

Syntax component	Meaning
Syntax element	**TOOL CALL 5 Z S3200 F300** Syntax elements are all parts of the NC function, such as technology values **S3200** or coordinate information. NC functions also contain optional syntax elements. The control shows certain syntax elements in color in the **Program** workspace. **Further information:** "Appearance of the NC program", Page 196
Value	**3200** for spindle speed **S** Not every syntax element must contain a numerical value, such as tool axis **Z**.

If you create NC programs in a text editor or outside of the control, note the correct spelling and sequence of the syntax elements.

Notes

NOTICE

Danger of collision!

With NC software 81762x-16, the TNC7 does not support ISO programming. There is a risk of collision during execution due to the missing support.

- Use Klartext NC programs exclusively.

- NC functions can also consist of more than one NC block, such as **BLK FORM**.
- Miscellaneous functions **M** and comments can be both syntax elements within NC functions as well as their own NC functions.
- Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.
- The file name extension ***.h** designates a Klartext program.

  Further information: "Programming fundamentals", Page 190

7.3.2 Editor operating mode

Application

In the **Editor** operating mode you can do the following:

- Create, edit and simulate NC programs
- Create and edit contours
- Create and edit pallet tables

Description of function

With **Add** you can create a new file or open an existing one. The control displays up to ten tabs.

The **Editor** presents the following workspaces if an NC program is open:

- **Help**

  Further information: "Help workspace", Page 1378
- **Contour**

  Further information: "Graphical Programming", Page 1337
- **Program**

  Further information: "Program workspace", Page 194
- **Simulation**

  Further information: "Simulation Workspace", Page 1405
- **Simulation status**

  Further information: "Simulation status workspace", Page 165
- **Keyboard**

  Further information: "Virtual keyboard of the control bar", Page 1380

If you open a pallet table, the control displays the workspaces **Job list** and **Form** for pallets. You cannot edit these workspaces.

Further information: "Job list workspace", Page 1764

Further information: "Form workspace for pallets", Page 1771

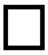
If option 154 is active, **Batch Process Manager** lets you use the complete scope of functions when executing pallet tables.

Further information: "Job list workspace", Page 1764

If an NC program or pallet table selected is in the **Program Run** operating mode, the controls shows the **M** status on the tab of the NC program. If the **Simulation** workspace for this NC program is open, the controls shows the **Control-in-operation** icon on the tab of the NC program.

Icons and buttons

The **Editor** operating mode contains the following icons and buttons:

Icon or button	Meaning
	The control uses this icon to show that an NC program is open.
	The control uses this icon to show that a contour is open. **Further information:** "Graphical Programming", Page 1337
	The control uses this icon to show that a pallet table is open. **Further information:** "Pallet Machining and Job Lists", Page 1763
Klartext programming	If this switch is active, then you are using dialog-guided programming. If this switch is not active, then you are programming in the text editor. **Further information:** "Editing NC programs", Page 202
Insert NC function	The control opens the **Insert NC function** window. **Further information:** "Editing NC programs", Page 202
GOTO block number	The control selects the block number that you defined. **Further information:** "GOTO function", Page 1383
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** "Q parameter list window", Page 1271
/ Skip Off/On	Mark NC blocks with a **/** character. NC blocks marked with a **/** character will be ignored during program run as soon as the **Skip /** switch is active. **Further information:** "Hiding NC blocks", Page 1385
; Comment Off/On	Insert or remove a **;** character in front of an NC block. If an NC block begins with a **;** character, then the block is a comment. **Further information:** "Adding comments", Page 1384
Edit	The control opens the context menu. **Further information:** "Context menu", Page 1392
Select in Program Run	The control opens the file in the **Program Run** operating mode. **Further information:** "Program Run", Page 1777
Start the simulation	The control opens the **Simulation** workspace and starts graphic simulation. **Further information:** "Simulation Workspace", Page 1405

7.3.3 Program workspace

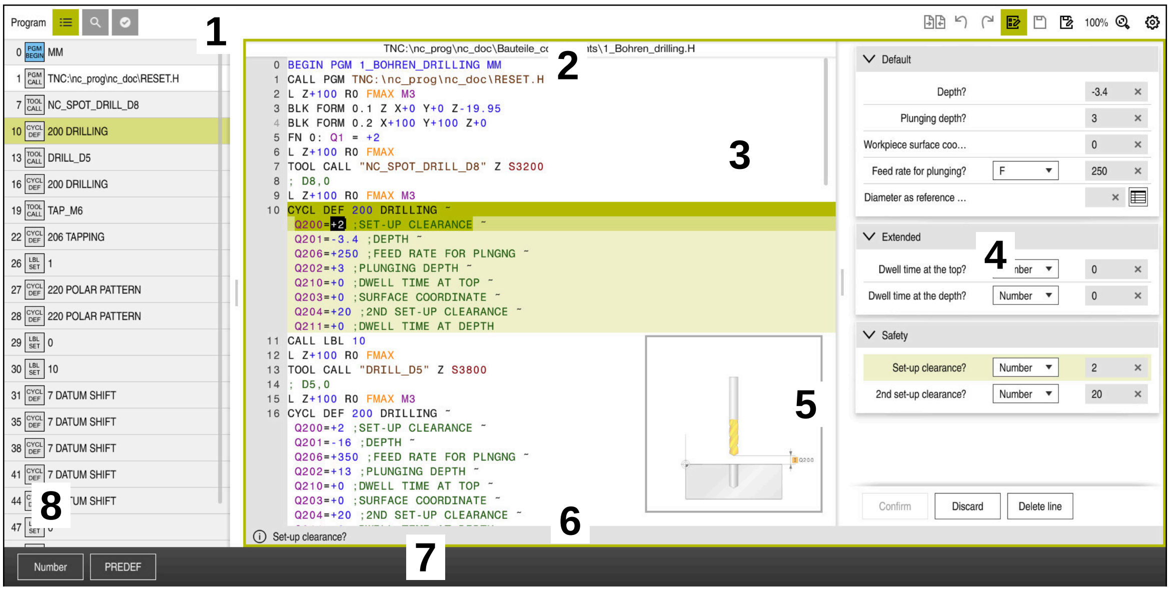
Application

The control displays the NC program in the **Program** workspace.

You can edit the NC program in the **Editor** operating mode and in the **MDI** application, but not in the **Program Run** operating mode.

Description of function

Areas of the Program workspace



Program workspace with active structure, help graphic and form

1 Title bar
 Further information: "Icons in the title bar", Page 196
2 File information bar
 In the file information bar, the control shows the path and file name of the NC program.
3 Contents of the NC program
 Further information: "Appearance of the NC program", Page 196
4 **Form** column
 Further information: "Form column in the Program workspace", Page 201
5 Help graphic of the syntax element being edited
 Further information: "Help graphic", Page 197
6 Dialog bar
 In the dialog bar, the control shows additional information or instructions for the syntax element being edited.
7 Action bar
 In the action bar, the control shows selection options for the syntax element being edited.
8 **Structure**, **Search** or **Tool check** column
 Further information: "Structure column in the Program workspace", Page 1386
 Further information: "Search column in the Program workspace", Page 1388
 Further information: "Tool usage test", Page 284

Icons in the title bar

The following icons are shown in the **Program** workspace in the title bar:

Further information: "Icons on the control's user interface", Page 102

Icon or shortcut	Function
	Open and close the **Structure** column **Further information:** "Structure column in the Program workspace", Page 1386
CTRL+F	Open and close the **Search** column **Further information:** "Search column in the Program workspace", Page 1388
	Open and close the **Tool check** column **Further information:** "Tool usage test", Page 284
	Activate and end comparison functions **Further information:** "Program comparison", Page 1391
	Show and hide the **Form** column **Further information:** "Form column in the Program workspace", Page 201
100%	Font size of the NC program When you select the percent value, the control will display icons for increasing and decreasing the font size.
	Set font size of the NC program to 100%
	Open the **Program settings** window **Further information:** "Settings in the Program workspace", Page 197

Appearance of the NC program

By default, the control displays the syntax in black color. The control displays the following syntax elements in color within the NC program:

Color	Syntax element
Brown	Text entries, e.g. tool name or file name
Blue	■ Numerical values ■ Structure items and texts
Dark green	Comments
Purple	■ Variables ■ Miscellaneous functions **M**
Dark red	■ Definition of spindle speed ■ Definition of feed rate
Orange	Rapid traverse **FMAX**

Help graphic

When you are editing an NC block, for some NC functions the control shows a help graphic for the current syntax element. The size of the help graphic depends on the size of the **Program** workspace.

The control shows the help graphic at the right edge of the workspace, or at the top or bottom edge. The help graphic is positioned in the half that does not contain the cursor.

When you tap or click the help graphic, the control will maximize the help graphic. If the **Help** workspace is open, the control displays the help graphic there.

Further information: "Help workspace", Page 1378

Settings in the Program workspace

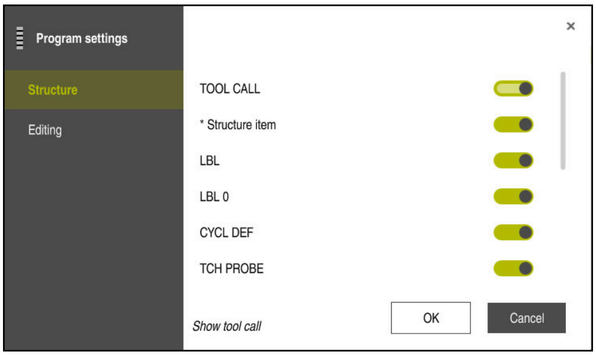
In the **Program settings** window you can influence contents shown in the **Program** workspace as well as the control's behavior there. The selected settings are modally effective.

The settings available in the **Program settings** window depend on the operating mode.

The **Program settings** window consists of the following areas:

- **Structure**
- **Editing**

Structure area



Structure area in the **Program settings** window

In the **Structure** area you use switches to choose which structure elements the control should display in the **Structure** column.

Further information: "Structure column in the Program workspace", Page 1386

The following structure elements are available:

- **TOOL CALL**
- *** Structure block**
- **LBL**
- **LBL 0**
- **CYCL DEF**
- **TCH PROBE**
- **MONITORING SECTION START**
- **MONITORING SECTION STOP**
- **PGM CALL**
- **FUNCTION MODE**
- **M30 / M2**
- **M1**
- **M0 / STOP**

Editing area

The **Editing** area contains the following settings:

Setting	Meaning
Automatic saving	**Save changes to the NC program automatically or manually** If the switch is active, the control saves the NC program automatically upon the following actions: ■ Switching between tabs ■ Starting the simulation ■ Closing the NC program ■ Switching the operating mode If the switch is not active, you must save manually. Upon the stated actions, the control asks whether the changes should be saved.
Allow syntax errors in text mode	If the switch is active, the control can save NC blocks in the text editor even if they contain syntax errors. If the switch is not active, you must correct all syntax errors within an NC block. Otherwise you cannot save the NC block. **Further information:** "Editing NC functions", Page 205
Generate absolute paths	**Create relative or absolute path entries** If the switch is active, the control uses absolute paths for called files, e.g.: **TNC:\nc_prog\$mdi.h**. If the switch is not active, the control uses relative paths , e.g.: **demo\reset.H**. If the file is located at a higher level in the folder structure than the calling NC program, the control creates an absolute path. **Further information:** "Path", Page 1070
Always save formatted	**Format NC program while saving** If an NC program has fewer than 30 000 characters, the control always formats the file when saving it, e.g.: capital letters for all syntax initiators. If the switch is active, the control also formats NC programs with more than 30 000 characters each time it saves the file. This can increase the time needed for saving. If the switch is not active, the control does not format NC programs with more than 30 000 characters.

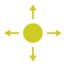
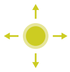
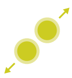
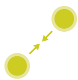
Using the Program workspace

The **Program** workspace can be used as follows:

- Touch operation
- Operation with keys and buttons
- Operation with a mouse

Touch operation

You use gestures to perform the following functions:

Symbol	Gesture	Meaning
	Tap	■ Select an NC block ■ Select a syntax element while editing
	Double tap	Edit an NC block
	Long press	Open the context menu If you are working with a mouse, click with the right mouse key. **Further information:** "Context menu", Page 1392
	Swipe	Scroll in an NC program
	Drag	Change the area in which NC blocks are marked. **Further information:** "Context menu in the Program workspace", Page 1395
	Spread	Increase the syntax font size
	Pinch	Reduce the syntax font size

Keys and buttons

You use keys and buttons to perform the following functions:

Key or button		Function
▲	▼	■ Navigate between NC blocks ■ During editing, search for the same syntax element in the NC program **Further information:** "Searching for the same syntax elements in different NC blocks", Page 201
▶	◀	■ Edit an NC block ■ During editing, navigate to previous or next syntax element
CTRL+ ▶	CTRL+ ◀	Navigate one position to the right or left within the value of a syntax element
GOTO		■ Use the block number to select an NC block directly **Further information:** "GOTO function", Page 1383 ■ Open selection menus during editing
⌖		Open position display of the control bar in order to copy the position If you select a line in the position display, the control copies the current value of this line to an open dialog.
CE		Delete value of a syntax element
NO ENT		Skip or remove optional syntax elements during programming
DEL BLK		Delete an NC block or cancel a dialog
END BLK		■ Confirm entry and conclude an NC block ■ Open the **Add** tab
ESC		Cancel editing without applying changes
Klartext progra...		Select **Klartext programming** mode or a text editor **Further information:** "Editing NC functions", Page 205
Insert NC function		Open the **Insert NC function** window **Further information:** "Inserting NC functions", Page 203
Edit		Open the context menu **Further information:** "Context menu", Page 1392

Searching for the same syntax elements in different NC blocks

If you are editing an NC block, you can search for the same syntax element in the rest of the NC program.

To search for a syntax element in the NC program:

- Select an NC block
- Edit the NC block
- Navigate to the desired syntax element
- Press the arrow up or down key
- > The control marks the next NC block that contains the syntax element. The cursor is on the same syntax element as in the previous NC block. Press the arrow up key to search backwards.

Notes

- If you search for the same syntax element in a very long NC program, the control displays a pop-up window. You can cancel the search at any time.
- Use the optional machine parameter **maxLineCommandSrch** (no. 105412) to define how many NC blocks the control searches for the same syntax element.
- When you open an NC program, the control checks whether the NC program is complete and syntactically correct.

  Use the optional machine parameter **maxLineGeoSrch**(no. 105408) to define up to which NC block the control should check the program.
- When you open an NC program without content, you can edit the **BEGIN PGM** and **END PGM** NC blocks and change the unit of measure of the NC program.
- An NC program is incomplete without the **END PGM** NC block.

  If you open an incomplete NC program in the **Editor** operating mode, the control automatically inserts the NC block.
- You cannot edit an NC program in the **Editor** operating mode if this NC program is being executed in the **Program Run** operating mode.

Form column in the Program workspace

Application

In the **Form** column in the **Program** workspace, the control shows all possible syntax elements for the currently selected NC function. You can edit all syntax elements in the form.

Related topics

- **Form** workspace for pallet tables

  Further information: "Form workspace for pallets", Page 1771
- Editing an NC function in the **Form** column

  Further information: "Editing NC functions", Page 205

Requirement

- **Klartext programming** mode must be active

Description of function

The control offers the following icons and buttons for using the **Form** column:

Icon or button	Function
	Show and hide the **Form** column
Confirm	Confirm entry and conclude an NC block
Discard	Discard entries and conclude an NC block
Delete line	Delete NC block

The control groups the syntax elements in the form depending on their functions, such as coordinates or safety.

The control indicates the required syntax elements with a red frame. Only once you have defined all of the required syntax elements can you confirm the entries and conclude the NC block. The control highlights the syntax element currently being edited.

If an input is invalid, the control displays an information symbol ahead of the syntax element. When you select the information symbol, the control displays information on the error.

Notes

- In the following cases the control shows no contents in the form:
  - NC program is being run
  - NC blocks are being marked
  - NC block contains syntax error(s)
  - **BEGIN PGM** or **END PGM** NC blocks are selected
- If you define more than one miscellaneous function in an NC block, you can use the arrows in the form to change the sequence of the miscellaneous functions.
- If you define a label with a number, the control shows an icon next to the input area. The control uses this symbol to assign the next available number to the label.

7.3.4 Editing NC programs

Application

The editing of NC programs refers both to the insertion of NC functions as well as their modification. You can also edit NC programs that you previously generated with a CAM system and then transmitted to the control.

Related topics

- Using the **Program** workspace

  Further information: "Using the Program workspace", Page 198

Requirements

You can edit NC programs only in the **Programming** operating mode and in the **MDI** application.

> In the **MDI** application you edit only the NC program **$mdi.h** or **$mdi_inch.h**.

Description of function

Inserting NC functions

Inserting an NC function directly with keys or buttons

You can directly insert frequently needed NC functions, such as path functions, with keys.

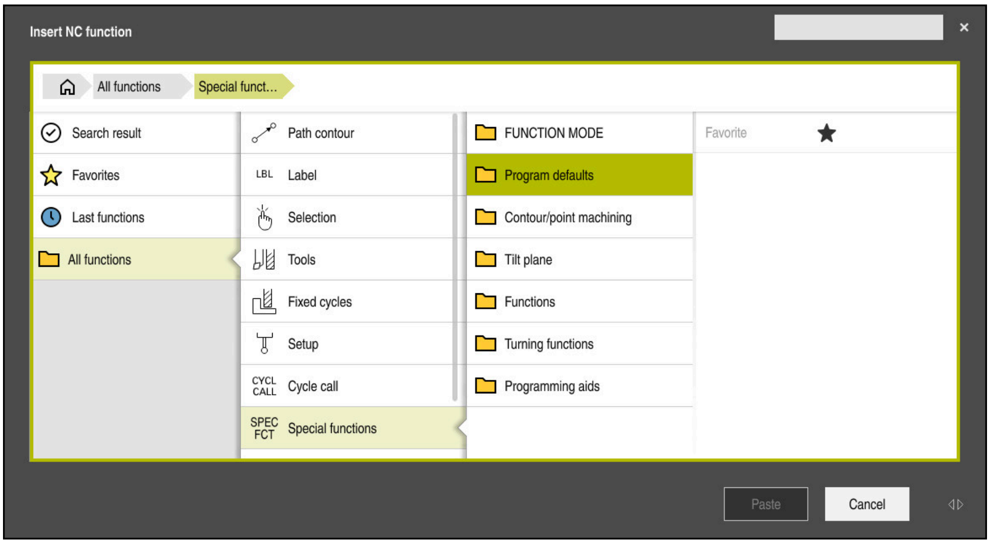
As an alternative to the keys, the control offers both the screen keyboard as well as the **Keyboard** workspace in NC input mode.

Further information: "Virtual keyboard of the control bar", Page 1380

To insert frequently needed NC functions:

- Select **L**
- > The control creates a new NC block and starts the dialog.
- Follow the instructions in the dialog

Inserting an NC function through selection



Insert NC function window

You can select all NC functions through the **Insert NC function** window.

You can navigate through the **Insert NC function** window as follows:

- Navigate through the tree structure manually, starting from **All functions**
- Use keys or buttons to narrow down the selection possibilities, e.g.: **CYCL DEF** key opens the cycle groups

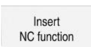
  Further information: "Keycaps for NC dialog", Page 99
- Ten most recently used NC functions under **Last functions**
- NC functions marked as favorites under **Favorites**

  Further information: "Icons on the control's user interface", Page 102
- Enter a search term under **Search for NC functions**

  The control displays the results under **Search result**.

> You can begin the search as soon as the **Insert NC function** window opens by entering a character.

To insert a new NC function:



- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Navigate to the desired NC function
- > The control highlights the selected NC function.

- ▶ Select **Paste**
- > The control creates a new NC block and starts the dialog.
- ▶ Follow the instructions in the dialog

Editing NC functions

Editing an NC function in the Klartext programming mode

By default, the control opens newly created and syntactically correct NC programs in the **Klartext programming** mode.

To edit an existing NC function in the **Klartext programming** mode:

- Navigate to the desired NC function
- Navigate to the desired syntax element
- > The control displays alternative syntax elements in the action bar.
- Select a syntax element
- Define a value, if necessary

END BLK

- Conclude entry, e.g. by pressing **END**

Editing an NC function in the Form column

If the **Klartext programming** mode is active, you can also use the **Form** column.

The **Form** does not just show the syntax elements selected and used, but rather all those that are possible for the current NC function.

To edit an existing NC function in the **Form** column:

- Navigate to the desired NC function

- Show the **Form** column
- Select an alternative syntax element if necessary, e.g:. **LP** instead of **L**
- If necessary, edit or add the value
- If necessary, enter an optional syntax element or select from a list, e.g.: miscellaneous function **M8**

Confirm

- Conclude entry, e.g. by pressing **Confirm**

Editing an NC function in the text editor mode

The control tries to correct syntax errors in the NC program automatically. If automatic correction is not possible, the control switches to text editor mode while editing this NC block. You must correct all errors before you can switch to **Klartext programming** mode.

> - If text editor mode is active, the **Klartext programming** switch is to the left and gray.
> - If you are editing an NC block with syntax errors, the only way to cancel editing is to press the **ESC** key.

To edit an existing NC function in the text editor mode

> The control underscores the faulty syntax element with a jagged red line and shows an information symbol before the NC function, e.g: for **FMX** instead of **FMAX**.

- Navigate to the desired NC function
- Select the information symbol
- > The control might open the **NC block auto-correction** window with a solution proposal.

Yes

- Apply the proposal to the NC program with **Yes** or cancel auto-correction.

> - The control cannot offer solution proposals in all cases.
> - The text editor mode supports all navigation possibilities of the **Program** workspace. But you can work more quickly in the text editor mode by using gestures or a mouse, since then you can select the information symbol directly, for example.

Notes

NOTICE

Danger of collision!

With NC software 81762x-16, the TNC7 does not support ISO programming. There is a risk of collision during execution due to the missing support.

- Use Klartext NC programs exclusively.

- The instructions include emphasized text strings, e.g: **200 DRILLING**. You can use these text strings for better searching in the **Insert NC function** window.
- When you are editing an NC function, use the arrows to navigate left and right to the syntax elements, even within cycles. The up and down arrows search for the same syntax element in the rest of the NC program.

  Further information: "Searching for the same syntax elements in different NC blocks", Page 201
- If you are editing an NC block and haven't saved yet, the **Undo** and **Redo** functions affect the individual syntax elements of the NC function.

  Further information: "Icons on the control's user interface", Page 102
- Press the **actual-position-capture** key for the control to open the position display of the status overview. You can copy the current value of an axis into the programming dialog.

  Further information: "Status overview on the control bar", Page 147
- Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.
- You cannot edit an NC program in the **Editor** operating mode if this NC program is being executed in the **Program Run** operating mode.

8

Technology-Specific NC Programming

8.1 Switching the operating mode with FUNCTION MODE

Application

The control offers a **FUNCTION MODE** operating mode for each of the technologies milling, milling-turning and grinding. Additionally, you can use **FUNCTION MODE SET** to activate settings defined by the machine manufacturer, e.g.: switching the traverse range.

Related topics

- Milling-turning operations (option 50)
  Further information: "Turning (option 50)", Page 212
- Grinding operations (option 156)
  Further information: "Grinding operations (option 156)", Page 224
- Editing kinematic models in the **Settings** application
  Further information: "Channel settings", Page 1915

Requirements

- Control adapted by the machine manufacturer
  The machine manufacturer defines which internal functions the control performs with this function. The machine manufacturer must define selection possibilities for the **FUNCTION MODE SET** function.
- For **FUNCTION MODE TURN**: software option 50 (Milling/turning)
- For **FUNCTION MODE GRIND**: software option 156 (Jig grinding)

Description of function

When the operating modes are switched, the control executes a macro that defines the machine-specific settings for the specific operating mode. With the NC functions **FUNCTION MODE TURN** and **FUNCTION MODE MILL**, you can activate a machine kinematic model that the machine manufacturer has defined and saved in the macro.

If the machine manufacturer has enabled the selection of various kinematic models, then you can switch between them using the **FUNCTION MODE** function.

If turning mode is active, the control shows a corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 141

Input

12 FUNCTION MODE TURN "AC_TURN"	; Activate turning mode with the selected kinematic model
11 FUNCTION MODE SET "Range1"	; Activate the machine manufacturer setting

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION MODE	Syntax initiator for the machining mode
MILL, **TURN**, **GRIND** or **SET**	Select the machining mode or machine manufacturer setting
" " or **QS**	Name of a kinematic model or machine manufacturer setting or QS parameter with the name You use choose the setting from a selection menu. Optional syntax element

Notes

WARNING

Caution: Significant property damage!

Very high physical forces are generated during turning, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!

- Clamp the workpiece in the spindle center
- Clamp workpiece securely
- Program low spindle speeds (increase as required)
- Limit the spindle speed (increase as required)
- Eliminate unbalance (calibrate)

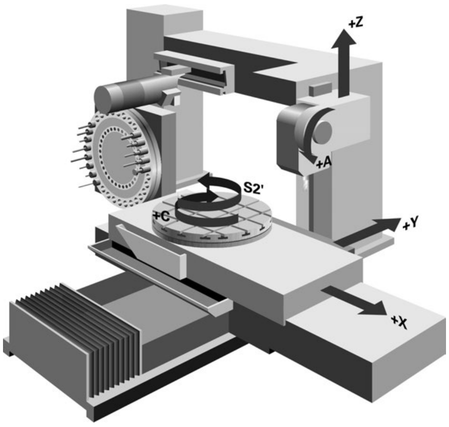
- In the optional machine parameter **CfgModeSelect**(no. 132200), the machine manufacturer defines the settings for the **FUNCTION MODE SET** function. If the machine manufacturer does not define the machine parameter, then **FUNCTION MODE SET** is not available.
- If the **Tilt working plane** or **TCPM** functions are active, you cannot switch the operating mode.
- The preset must be in the center of the turning spindle in turning mode.

8.2 Turning (option 50)

8.2.1 Fundamentals

Depending on the machine and kinematics, it is possible to perform both milling and turning operations on milling machines. A workpiece can thus be machined completely on one machine, even if complex milling and turning applications are required.

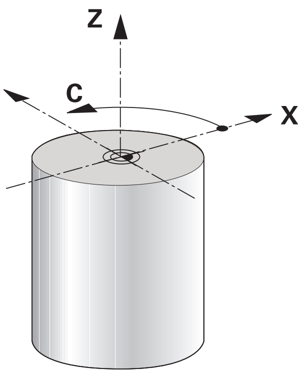
In a turning operation, the tool is in a fixed position, whereas the rotary table and the clamped workpiece rotate.



NC fundamentals for turning

The assignment of the axes with turning is defined so that the X coordinates describe the diameter of the workpiece and the Z coordinates the longitudinal positions.

Machining is thus always done in the **ZX** working plane. The machine axes to be used for the required movements depend on the respective machine kinematics and are determined by the machine manufacturer. This makes NC programs with turning functions largely exchangeable and independent of the machine model.



Workpiece preset for turning operations

On the control, you can simply switch between milling and turning mode within your NC program. In turning mode, the rotary table serves as lathe spindle, whereas the milling spindle with the tool is fixed. This way, it is possible to machine rotationally symmetric contours. The tool reference point must always be at the center of the lathe spindle.

Further information: "Preset management", Page 949

If you use a facing slide, you can set the workpiece preset to a different location, since in this case the tool spindle performs the turning operation.

Further information: "Using the facing head with FACING HEAD POS (option 50)", Page 1199

Production processes

Depending on the machining direction and task, turning applications can be subdivided into different production processes, e.g.:

- Longitudinal turning
- Face turning
- Recess turning
- Thread cutting

The control provides several cycles for each of the various production processes.

Further information: "Cycles for milling and turning", Page 677

You can run the cycles with an inclined tool in order to produce undercuts.

Further information: "Inclined turning", Page 216

Tools for turning operations

When managing turning tools, other geometric descriptions than those for milling or drilling tools are required. To execute a tool-tip radius compensation, for example, the definition of the cutting-edge radius is required. The control provides a special tool table for turning tools. In tool management, the control displays only the required tool data for the current tool type.

Further information: "Tool data", Page 249

Further information: "Tooth radius compensation for turning tools (option 50)", Page 1041

You can correct turning tool values in the NC program.

The control offers the following functions for this:

- Cutter radius compensation

  Further information: "Tooth radius compensation for turning tools (option 50)", Page 1041
- Compensation Tables

  Further information: "Tool compensation with compensation tables", Page 1044
- The **FUNCTION TURNDATA CORR** function

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1047

Notes

⚠ WARNING

Caution: Significant property damage!

Very high physical forces are generated during turning, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!

- Clamp the workpiece in the spindle center
- Clamp workpiece securely
- Program low spindle speeds (increase as required)
- Limit the spindle speed (increase as required)
- Eliminate unbalance (calibrate)

- The orientation of the tool spindle (spindle angle) depends on the machining direction. The tool tip is aligned to the center of the turning spindle for outside machining. For inside machining, the tool points away from the center of the turning spindle.

  The direction of spindle rotation must be adapted when the machining direction (outside/inside machining) is changed.

  Further information: "Overview of miscellaneous functions", Page 1225
- During turning, the cutting edge and the center of the turning spindle must be at the same level. During turning, the tool therefore has to be pre-positioned to the Y coordinate of the turning-spindle center.
- In turning mode, diameter values are displayed on the X axis position display. The control then shows an additional diameter symbol.

  Further information: "Positions workspace", Page 141
- In turning mode, the spindle potentiometer is active for the turning spindle (rotary table).
- In turning mode, no coordinate conversion cycles are permitted except for the datum shift.

  Further information: "Datum shift with TRANS DATUM", Page 972
- In turning mode, the **SPA**, **SPB** and **SPC** transformations from the preset table are not permitted. If you activate one of these transformations while executing the NC program in turning mode, the control will display the **Transformation not possible** error message.
- The machining times determined using the graphic simulation do not correspond to the actual machining times. Reasons for this during combined milling-turning operations include the switching of operating modes.

  Further information: "Simulation Workspace", Page 1405

8.2.2 Technology values for turning operations

Defining the spindle speed for turning with FUNCTION TURNDATA SPIN

Application

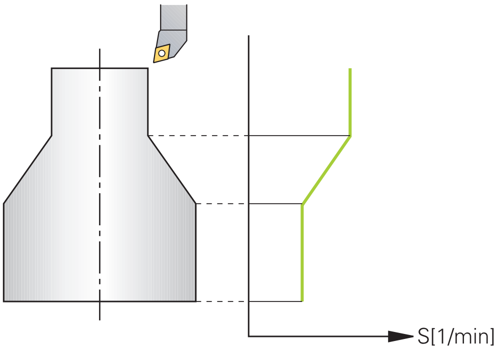
With turning you can machine both at constant spindle speed and constant cutting speed.

Use **FUNCTION TURNDATA SPIN** to define the speed.

Requirement

- Machine with at least two rotary axes
- Combined milling/turning (software option 50)

Description of function



If you machine at constant cutting speed **VCONST:ON**, the control modifies the speed according to the distance of the tool tip to the center of the turning spindle. For positioning movements toward the center of rotation, the control increases the table speed; for movements away from the center of rotation, it reduces the table speed.

For processing with constant spindle speed **VCONST:Off**, speed is independent of the tool position.

With **FUNCTION TURNDATA SPIN** you can define a maximum speed for the constant speed.

Input

11 FUNCTION TURNDATA SPIN VCONST:ON VC:100 GEARRANGE:2	; Constant surface speed with gear range 2

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION TURNDATA SPIN	Syntax initiator for speed definition in turning mode
VCONST OFF or **ON**	Definition of a constant cutting speed or constant surface speed Optional syntax element
VC	Value for the surface speed Optional syntax element
S or **SMAX**	Constant speed or speed limitation Optional syntax element
GEARRANGE	Gear range for the lathe spindle Optional syntax element

Notes

- If you machine at constant cutting speed, the selected gear range limits the possible spindle speed range. The possible gear ranges (if applicable) depend on your machine.
- When the maximum speed has been reached, the control displays **SMAX** instead of **S** in the status display.
- To reset the speed limitation, program **FUNCTION TURNDATA SPIN SMAX0**.
- In turning mode, the spindle potentiometer is active for the turning spindle (rotary table).
- Cycle **800** limits the maximum spindle speed during eccentric turning. The control restores a programmed limitation of the spindle speed after eccentric turning.

  Further information: "Cycle 800 ADJUST XZ SYSTEM ", Page 681

Feed rate

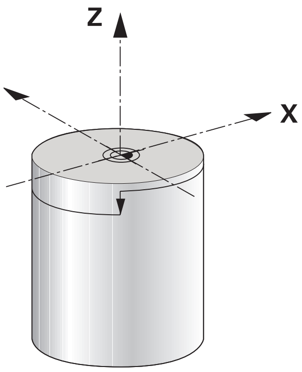
Application

With turning, feed rates are often specified in millimeters per revolution. Use the miscellaneous function **M136** for this on the control.

Further information: "Interpreting the feed rate as mm/rev with M136", Page 1250

Description of function

With turning, feed rates are often specified in millimeters per revolution. The control thus moves the tool at a defined value for every spindle rotation. The resulting contouring feed rate is thus dependent on the speed of the turning spindle. The control increases the feed rate at high spindle speeds and reduces it at low spindle speeds. This enables you to machine with uniform cutting depth and constant cutting force, thus achieving constant chip thickness



Note

During many turning operations, it is not possible to maintain constant surface speeds (**VCONST: ON**) because the maximum spindle speed is reached first. Use the machine parameter **facMinFeedTurnSMAX** (no. 201009) to define the behavior of the control after the maximum speed has been reached.

8.2.3 Inclined turning

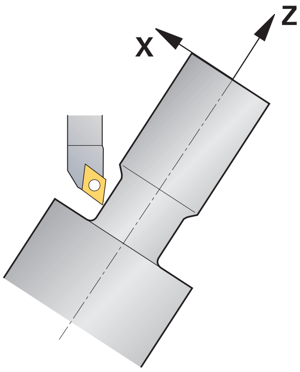
Application

It may sometimes be necessary for you to bring the swivel axes into a specific position to machine a specific process. This can be necessary, for example, when you can only machine contour elements according to a specific position due to tool geometry.

Requirement

- Machine with at least two rotary axes
- Combined milling/turning (software option 50)

Description of function



The control offers the following methods of inclined turning:

Function	Description	Further information
M144	The control uses **M144** in subsequent traverse movements to compensate for tool offsets that result from inclined rotary axes.	Page 1255
M128	With **M128** the control behaves like with **M144**, but you cannot use cutter radius compensation outside of cycles.	Page 1245
FUNCTION TCPM with **REFPNT TIP-CENTER**	Use **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** to activate the virtual tool tip. If you activate inclined machining with **FUNCTION TCPM** with **REFPNT TIP-CENTER**, then tool-tip radius compensation is also possible without a cycle; that is, in traversing blocks with **RL**/**RR**. HEIDENHAIN recommends using **FUNCTION TCPM** with **REFPNT TIP-CENTER**.	Page 1027
Cycle **800**	Use Cycle **800 ADJUST XZ SYSTEM** to define an inclination angle.	Page 681

If you execute turning cycles with **M144**, **FUNCTION TCPM**, or **M128**, then the angles of the tool relative to the contour will change. The control automatically takes these modifications into account and therefore also monitors the inclined machining operation.

Notes

- Threading cycles can be run with inclined machining only if the tool is at a right angle (+90°, or –90°).
- Tool compensation **FUNCTION TURNDATA CORR-TCS** is always effective in the tool coordinate system, even during inclined machining.

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1047

8.2.4 Simultaneous turning

Application

You can combine the turning operation with function **M128** or **FUNCTION TCPM** and **REFPNT TIP-CENTER**. This enables you to manufacture contours in one cut, for which you have to change the inclination angle (simultaneous machining).

Related topics

- Cycles for simultaneous turning (option 158)

  Further information: "Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING (option158) ", Page 816
- Miscellaneous function **M128** (option 9)

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1245
- **FUNCTION TCPM** (option 9)

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027

Requirements

- Machine with at least two rotary axes
- Combined milling/turning (software option 50)
- Software option 9: Advanced Functions (set 2)

Description of function

The simultaneous turning contour is a turning contour for which a rotary axis whose inclination does not violate the contour can be programmed on **CP** polar circles and **L** linear blocks. Collisions with lateral cutting edges or holders are not prevented. This makes it possible to finish contours with one tool in a continuous movement, even though different sections of the contour are accessible only in different tool inclinations.

In the NC program you define how the rotary axis has to be inclined to reach the different contour parts without collisions.

Use the cutter radius oversize **DRS** to leave an equidistant oversize on the contour.

Use **FUNCTION TCPM** and **REFPNT TIP-CENTER** to measure the theoretical tool tip of the turning tools being used for this.

The following requirements apply if you want to use **M128** for simultaneous turning:

- Only for NC programs programmed on the path of the tool center.
- Only for button turning tools with TO 9

  Further information: "Subgroups of technology-specific tool types", Page 256
- The tool must be measured at the center of the tool-tip radius

Further information: "Presets on the tool", Page 244

Example

An NC program with simultaneous turning includes the following components:

- Activate turning mode
- Insert a turning tool
- Adjust the coordinate system with cycle **800 ADJUST XZ SYSTEM**
- Activate **FUNCTION TCPM** with **REFPNT TIP-CENTER**
- Activate cutter radius compensation with **RL/RR**
- Program simultaneous turning contour
- End cutter radius compensation with **R0** or by departing the contour
- Reset **FUNCTION TCPM**

0 BEGIN PGM TURNSIMULTAN MM	
*** - ...**	
12 FUNCTION MODE TURN	; Activate turning mode
13 TOOL CALL "TURN_FINISH"	; Insert turning tool
14 FUNCTION TURNDATA SPIN VCONST:OFF S500	
15 M140 MB MAX	
*** - ...**	; Adjust the coordinate system
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+90 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+0 ;INCLINED MACHINING ~	
Q531=+0 ;ANGLE OF INCIDENCE ~	
Q532= MAX ;FEED RATE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP	
17 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	; Activate **FUNCTION TCPM**
18 FUNCTION TURNDATA CORR-TCS:Z/X DRS:-0.1	
19 L X+100 Y+0 Z+10 R0 FMAX M304	
20 L X+45 RR FMAX	; Activate cutter radius compensation with **RR**
*** - ...**	
26 L Z-12.5 A-75	; Program simultaneous turning contour
27 L Z-15	
28 CC X+69 Z-20	
29 CP PA-90 A-45 DR-	
30 CP PA-180 A+0 DR-	
*** - ...**	
47 L X+100 Z-45 R0 FMAX	; End cutter radius compensation with **R0**
48 FUNCTION RESET TCPM	; Reset **FUNCTION TCPM**
49 FUNCTION MODE MILL	
*** - ...**	
71 END PGM TURNSIMULTAN MM	

8.2.5 Turning operation with FreeTurn tools

Application

The control makes it possible to define FreeTurn tools and to use them, e.g., for inclined or simultaneous turning operations.

FreeTurn tools are lathe tools that are equipped with multiple cutting edges. Depending on the variant, a single FreeTurn tool may be capable of axis-parallel and contour-parallel roughing and finishing.

Thanks to the use of FreeTurn tools, fewer tool changes are required, reducing the machining time. Due to the tool orientation to the workpiece, only outside machining is possible.

Related topics

- Inclined turning

  Further information: "Inclined turning", Page 216
- Simultaneous turning operation

  Further information: "Simultaneous turning", Page 218
- FreeTurn tools

  Further information: "Tool data", Page 249
- Indexed tools

  Further information: "Indexed tool", Page 250

Requirements

- Machine whose tool spindle is perpendicular to the workpiece spindle or can be inclined.

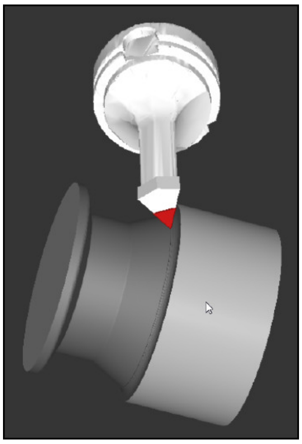
  Depending on the machine kinematics, a rotary axis is required for the orientation of the spindles to each other.
- Machine with controlled tool spindle

  The control inclines the cutting edge by means of inclining the tool spindle.
- Combined milling/turning (software option 50)
- Kinematics description

  The machine manufacturer provides the kinematics description. Based on the kinematics description, the control can take, e.g., the tool geometry into account.
- Machine-manufacturer macros for simultaneous turning with FreeTurn tools
- FreeTurn tool with suitable tool carrier
- Tool definition

  A FreeTurn tool always includes three cutting edges of an indexed tool.

Description of function

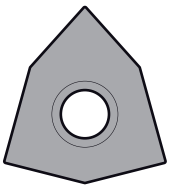


FreeTurn tool in simulation

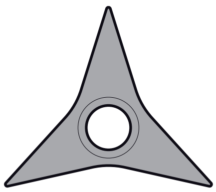
To use FreeTurn tools, call only the desired cutting edge of the correctly defined indexed tool in your NC program.

Further information: "Example: Turning with a FreeTurn tool", Page 833

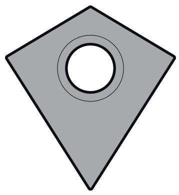
FreeTurn tools



FreeTurn indexable insert for roughing



FreeTurn indexable insert for finishing



FreeTurn indexable insert for roughing and finishing

The control supports all variants of FreeTurn tools:

- Tool with finishing cutting edge
- Tool with roughing cutting edge
- Tool with finishing and roughing cutting edge

In the **TYPE** column of the tool management, select a turning tool (**TURN**) as the tool type. In the **TYPE** column, assign the appropriate technology-specific tool type to each cutting edge, i.e. roughing tool (**ROUGH**) or finishing tool (**FINISH**).

Further information: "Subgroups of technology-specific tool types", Page 256

A FreeTurn tool must be defined as an indexed tool with three cutting edges that are offset by the **ORI** angle of orientation. Each cutting edge has the **TO 18** tool orientation.

Further information: "Example FreeTurn tool", Page 253

FreeTurn tool carrier

Tool carrier template for a FreeTurn tool

There is a suitable tool carrier for each FreeTurn tool variant. HEIDENHAIN provides ready-to-use tool carrier templates for download that are included in the programming station software. You can then assign the tool-carrier kinematics descriptions generated from the templates to the respective indexed cutting edge.

Further information: "Tool-carrier templates", Page 275

Notes

NOTICE
Danger of collision! The shaft length of the turning tool limits the diameter that can be machined. There is a risk of collision during machining! ▶ Check the machining sequence in the simulation

- Due to the tool orientation to the workpiece, only outside machining is possible.
- Please note that FreeTurn tools can be combined with various machining strategies. Therefore, make sure to observe the specific notes, e.g. in conjunction with the selected machining cycles.

8.2.6 Unbalance in turning operations

Application

In a turning operation, the tool is in a fixed position, whereas the rotary table and the clamped workpiece rotate. Depending on the size of the workpiece, the mass that is set in rotation can be very large. As the workpiece rotates, it creates an outward centrifugal force.

The control offers functions to detect the unbalance and support you in compensating for it.

Related topics

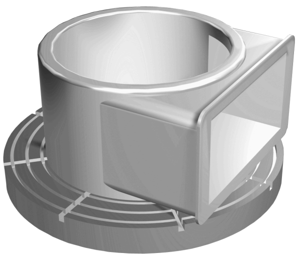
- Cycle **892 CHECK UNBALANCE**
  Further information: "Cycle 892 CHECK UNBALANCE ", Page 690
- Cycle **239 ASCERTAIN THE LOAD** (option 143)
  Further information: "Cycle 239 ASCERTAIN THE LOAD (option 143)", Page 1148

Description of function

Refer to your machine manual.

Unbalance functions are not required and available on all machine tool types.

The unbalance functions described here are basic functions that are set up and adapted to the machine by the machine manufacturer. The scope and effect of the described functions may therefore vary from machine to machine. The machine manufacturer may also provide different unbalance functions.



The centrifugal force that occurs basically depends on the rotational speed, the mass and the unbalance of the workpiece. A body with an uneven mass distribution that is put into rotary motion produces an unbalance. If the mass object is rotating, this creates outward-acting centrifugal forces. If the rotating mass is evenly distributed, the centrifugal forces cancel each other out. You compensate for the arising centrifugal forces by attaching balancing weights.

With Cycle **892 CHECK UNBALANCE** you define the maximum permissible unbalance and the maximum shaft speed. The control monitors these entries.

Further information: "Cycle 892 CHECK UNBALANCE ", Page 690

Unbalance monitor

The Unbalance Monitor function monitors the unbalance of a workpiece in turning mode. If a maximum unbalance limit specified by the machine manufacturer is exceeded, the control issues an error message and initiates an emergency stop.

In addition, you can further decrease the permissible unbalance limit by setting the optional machine parameter **limitUnbalanceUsr** (no. 120101). If this limit is exceeded, the control issues an error message. The control does not stop table rotation.

The control automatically activates the Unbalance Monitor function when you switch to turning mode. The unbalance monitor is effective until you switch back to milling mode.

Further information: "Switching the operating mode with FUNCTION MODE", Page 210

Notes

⚠ WARNING

Caution: Significant property damage!

Very high physical forces are generated during turning, for example due to high rotational speeds and heavy or unbalanced workpieces. Incorrect machining parameters, neglected unbalances or improper fixtures lead to an increased risk of accidents during machining!

- Clamp the workpiece in the spindle center
- Clamp workpiece securely
- Program low spindle speeds (increase as required)
- Limit the spindle speed (increase as required)
- Eliminate unbalance (calibrate)

- The rotation of the workpiece creates centrifugal forces that lead to vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life.
- The removal of material during machining will change the mass distribution within the workpiece. This generates the unbalance, which is why an unbalance test is recommended even between the machining steps.
- To compensate an unbalance, several balancing weights at different positions may be required.

8.3 Grinding operations (option 156)

8.3.1 Fundamentals

Special types of milling machines allow performing both milling and grinding operations. A workpiece can thus be machined completely on one machine, even if complex milling and grinding operations are required.

Requirements

- Software option 156 Jig grinding
- Available kinematics description for jig grinding

  The machine manufacturer creates the kinematics description.

Production processes

The term grinding encompasses many types of machining that differ in quite a few respects, e.g.:

- Jig grinding
- Cylindrical grinding
- Surface grinding

The TNC7 currently features jig grinding.

Jig grinding is the grinding of a 2D contour. The tool movement in the plane is optionally superimposed by a reciprocation movement along the active tool axis.

Further information: "Jig grinding", Page 226

If grinding is enabled on your milling machine, (Option 156), the dressing function is also available. This means that you can shape or resharpen the grinding wheel in the machine.

Further information: "Dressing", Page 227

Reciprocating stroke

For jig grinding, the movement of the tool in the plane can be superimposed by a stroke movement, the so-called reciprocating stroke. The superimposed stroke movement is effective in the active tool axis.

You define an upper and a lower stroke limit and can start and stop the reciprocating stroke and reset the corresponding values. The reciprocating stroke is effective until you stop it. **M2** or **M30** will stop the reciprocating stroke automatically.

The control provides cycles for defining, starting, and stopping reciprocating strokes.

As long as the reciprocating stroke is active in the program run, you cannot change to the other applications of the **Manual** operating mode.

The control presents the reciprocating stroke in the **Simulation** workspace of the **Program Run** operating mode.

Tools for grinding

When managing grinding tools, other geometric descriptions than those for milling or drilling tools are required. The control provides a special tool table for grinding and dressing tools. In tool management, the control displays only the required tool data for the current tool type.

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 1828

Further information: "Dressing tool table tooldress.drs (option 156)", Page 1836

You can use compensation tables to change the values of grinding tools during program run.

Further information: "Tool compensation with compensation tables", Page 1044

Structure of an NC program for grinding

An NC program for grinding is structured as follows:

- Dressing of the grinding tool, if required

  Further information: "General information on the dressing cycles", Page 843
- Defining the reciprocating stroke

  Further information: "Cycle 1000 DEFINE RECIP. STROKE (option 156)", Page 838
- If necessary, explicitly starting the reciprocating stroke

  Further information: "Cycle 1001 START RECIP. STROKE (option 156)", Page 840
- Moving along the contour
- Stopping the reciprocating stroke

  Further information: "Cycle 1002 STOP RECIP. STROKE (option 156)", Page 842

You can use specific machining cycles (e.g., cycles for grinding, for machining pockets or studs, or SL cycles) to define the contour.

Further information: "Cycles for grinding", Page 836

8.3.2 Jig grinding

Application

On a milling machine, jig grinding will mainly be used for finishing a pre-machined contour with a grinding tool. There is not much of a difference between jig grinding and milling. Instead of a milling cutter, a grinding tool is used, such as a grinding pin or a grinding wheel. Jig grinding produces more precise results and a better surface quality than milling.

Related topics

- Cycles for grinding

  Further information: "Cycles for grinding", Page 836
- Tool data for grinding tools

  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 1828
- Dressing of grinding tools

  Further information: "Dressing", Page 227

Requirements

- Software option 156 Jig grinding
- Available kinematics description for jig grinding

  The machine manufacturer creates the kinematics description.

Description of function

Machining is performed in milling mode, i.e. with **FUNCTION MODE MILL**.

Grinding cycles provide special movements for the grinding tool. A stroke or oscillating movement, the so-called reciprocating stroke, is superimposed with the movement in the working plane.

Grinding is also possible with a tilted working plane. The tool reciprocates along the active tool axis in the current working plane coordinate system (**WPL-CS**).

Notes

- The control does not support block scans while the reciprocating stroke is active.
  Further information: "Block scan for mid-program startup", Page 1786
- The reciprocating stroke continues to be effective during a programmed **STOP** or **M0** as well as in **Single Block** mode even after the end of an NC block.
- If no cycle has been programmed and a contour is being ground whose smallest inside radius is smaller than the tool radius, the control will display an error message.
- If you machine with SL cycles, only those areas will be ground that are suitable for the given tool radius. In this case, the resulting contour will not be completely finished and may need to be reworked.

8.3.3 Dressing

Application

The term “dressing” refers to the sharpening or trueing up of a grinding tool inside the machine. During dressing, the dresser machines the grinding wheel. Thus, in dressing, the grinding tool is the workpiece.

Related topics

- Activating dressing mode with **FUNCTION DRESS**
  Further information: "Activating dressing mode with FUNCTION DRESS", Page 228
- Cycles for dressing
  Further information: "General information on the dressing cycles", Page 843
- Tool data for dressing tools
  Further information: "Dressing tool table tooldress.drs (option 156)", Page 1836
- Jig grinding
  Further information: "Jig grinding", Page 226

Requirements

- Software option 156 Jig grinding
- Available kinematics description for jig grinding
  The machine manufacturer creates the kinematics description.

Description of function

The dresser removes material and thereby changes the dimensions of the grinding wheel. Dressing the diameter, for example, causes the radius of the grinding wheel to become smaller.

In dressing, the workpiece datum is located on an edge of the grinding wheel. Select the respective edge using Cycle **1030 ACTIVATE WHEEL EDGE**.

During dressing, the axes are arranged such that the X coordinates describe positions on the radius of the grinding wheel, and the Z coordinates describe the positions along the axis of the grinding wheel. The dressing programs are thus not contingent on the machine type.

The machine manufacturer defines which machine axes will perform the programmed movements.

Simplified dressing with a macro

Your machine tool builder can program the entire dressing mode in a macro.

In this case, the machine manufacturer determines the dressing sequence. It is not necessary to program **FUNCTION DRESS BEGIN**.

Depending on this macro, you can start the dressing mode with one of the following cycles:

- Cycle **1010 DRESSING DIAMETER**
- Cycle **1015 PROFILE DRESSING**
- Cycle **1016 DRESSING OF CUP WHEEL**
- OEM cycle

Notes

- For dressing operations, the machine must be prepared accordingly by the machine manufacturer. The machine manufacturer may provide his own cycles.
- Measure the grinding tool after dressing so that the control enters the correct delta values.
- Not all grinding tools require dressing. Comply with the information provided by your tool manufacturer.

8.3.4 Activating dressing mode with FUNCTION DRESS

Application

With **FUNCTION DRESS** you activate a dressing kinematic model for dressing a grinding tool. The grinding tool is then the workpiece and the axes may move in the opposite direction.

Your machine manufacturer might provide a simplified dressing procedure.

Further information: "Simplified dressing with a macro", Page 228

Related topics

- Cycles for dressing
  Further information: "General information on the dressing cycles", Page 843
- Fundamentals of dressing
  Further information: "Dressing", Page 227

Requirements

- Software option 156 Jig grinding
- Available kinematics description for dressing
  The machine manufacturer creates the kinematics description.
- Grinding tool is inserted
- Grinding tool without assigned tool-carrier kinematics

Description of function

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode in the **Program run, single block** or **Program run, full sequence** operating mode only
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine tool builder

For the control to switch to the kinematic model for dressing, you must program the dressing process between the functions **FUNCTION DRESS BEGIN** and **FUNCTION DRESS END**.

If dressing mode is active, the control shows a corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 141

You can switch back to normal operation with the function **FUNCTION DRESS END**.

In the event of an NC program abort or a power interruption, the control automatically activates normal operation and the kinematic model that was active prior to dressing mode.

Input

11 FUNCTION DRESS BEGIN "Dress"	; Activate dressing mode with the **Dress** kinematics

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION DRESS	Syntax initiator for dressing mode
BEGIN or **END**	Activate or deactivate dressing mode
Name or **QS**	Name of the selected kinematic model Fixed or variable name Only if **BEGIN** has been selected Optional syntax element

Notes

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

NOTICE

Danger of collision!

With an active kinematic model, the machine movements may be in the opposite direction. There is a risk of collision when moving the axes!

- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

- During dressing, the cutting edge of the dresser must be at the same height as the grinding wheel. The programmed Y coordinate must be 0.
- With the switch to dressing mode, the grinding tool remains in the spindle and retains its current rotational speed.
- The control does not support a block scan during the dressing process. If, during a block scan, you select the first NC block after the dressing operation, then the control moves to the most recently approached position in the dressing operation.

  Further information: "Block scan for mid-program startup", Page 1786
- If the “tilt working plane” function or **TCPM** function is active, then you cannot switch to dressing mode.
- The control resets the manual tilting functions (option 8) and **FUNCTION TCPM** (option 9) when dressing mode is activated.

  Further information: "3-D rotation window (option 8)", Page 1022
  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027
- In dressing mode you can use **TRANS DATUM** to change the workpiece datum. No other NC functions or coordinate conversion cycles are permitted in dressing mode. The control displays an error message.

  Further information: "Datum shift with TRANS DATUM", Page 972
- The **M140** function is not allowed in dressing mode. The control displays an error message.
- The control does not graphically depict the dressing operation. The times determined by the simulation do not reflect the actual machining times. One reason for this is the necessary switching of the kinematic model.

9

Workpiece Blank

9.1 Defining a workpiece blank with BLK FORM

Application

You use the **BLK FORM** function to define a workpiece blank for graphic simulation of the NC program.

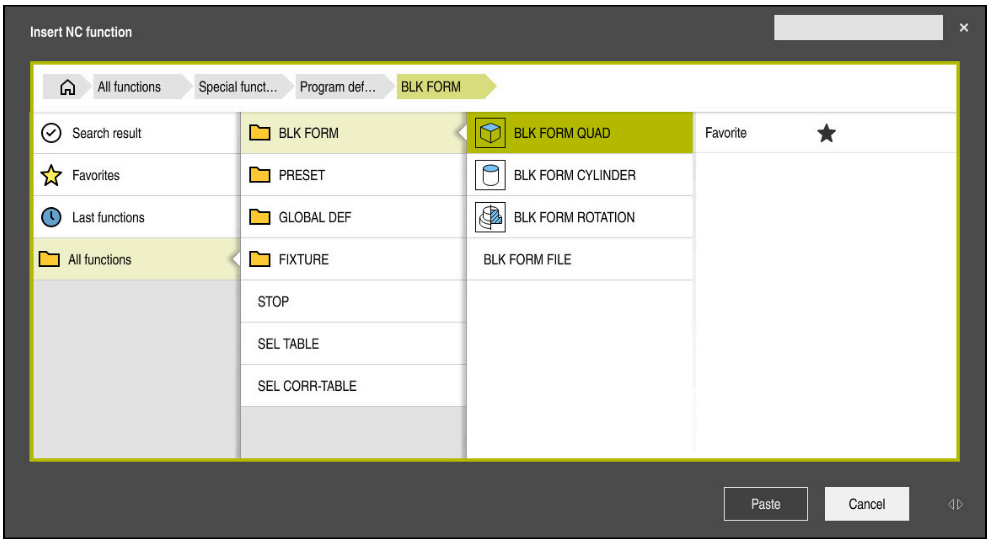
Related topics

- Depiction of the workpiece blank in the **Simulation** workspace
  Further information: "Simulation Workspace", Page 1405
- Blank form update with **FUNCTION TURNDATA BLANK** (option 50)
  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1047

Description of function

You define the blank relative to the workpiece preset.

Further information: "Presets in the machine", Page 187



The **Insert NC function** window for workpiece blank definition

When you create a new NC program, the control automatically opens the **Insert NC function** window for workpiece blank definition.

Further information: "Creating a new NC program", Page 112

The control offers the following workpiece blank definitions:

Icon	Function	Further information
	BLK FORM QUAD Cuboid workpiece blank	Page 236
	BLK FORM CYLINDER Cylindrical workpiece blank	Page 237
	BLK FORM ROTATION Rotationally symmetric blank with a definable contour	Page 238
	BLK FORM FILE STL file as workpiece blank and finished part	Page 239

Notes

NOTICE

Danger of collision!

Even if Dynamic Collision Monitoring (DCM) is active, the control does not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!

- Enable the **Advanced checks** switch for simulations
- Check the machining sequence using a simulation
- Carefully test your NC program or program section in the **Single Block** mode

- You can select files or subprograms in the following ways:
  - Enter the file path
  - Enter the number or name of the subprogram
  - Select a file or subprogram by means of a selection window
  - Define the file path or name of the subprogram in a QS parameter
  - Define the number of the subprogram in a Q, QL, or QR parameter

  If the called file is located in the same folder as the calling NC program, it is also possible to simply enter the file name.
- In order for the control to show the workpiece blank in the simulation, the workpiece blank must have minimum dimensions. The minimum dimensions are 0.1 mm or 0.004 inches in all axes as well as in the radius.
- The control shows the workpiece blank in the simulation only after it has executed the complete workpiece blank definition.
- Even if you want to close the **Insert NC function** window or add a workpiece blank definition after writing an NC program, you can always define a workpiece blank via the **Insert NC function** window.
- The **Advanced checks** function in the simulation uses the information from the workpiece blank definition for workpiece monitoring. The control can monitor only the active workpiece blank, even if several workpieces are clamped in the machine!

  Further information: "Advanced checks in the simulation", Page 1109
- In the **Simulation** workspace you can export the current view of the workpiece as an STL file. This function allows you to create missing 3-D models, for example semifinished parts if there are several machining steps.

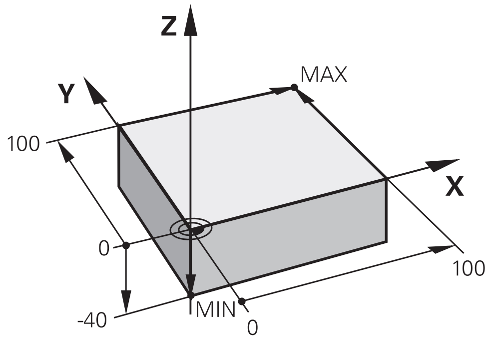
  Further information: "Exporting a simulated workpiece as STL file", Page 1415

9.1.1 Cuboid workpiece blank with BLK FORM QUAD

Application

With **BLK FORM QUAD** you define a cuboid workpiece blank. You use a MIN point and a MAX point to define a spatial diagonal.

Description of function



Cuboid workpiece blank with MIN point and MAX point

The sides of the cuboid are parallel to the **X**, **Y** and **Z** axes.

You define the cuboid by entering a MIN point for the bottom front left corner and a MAX point for the top rear right corner.

You define the coordinates of the points in the **X**, **Y** and **Z** relative to the workpiece preset. If you define a positive value for the MAX point in the Z coordinate, the blank is given an oversize.

Further information: "Presets in the machine", Page 187

If you use a cuboid workpiece blank for turning (option 50), keep the following in mind:

Even if the turning operation takes place in a two-dimensional plane (Z and X coordinates), you have to program the Y values for a rectangular blank in the definition of the workpiece blank.

Further information: "Fundamentals", Page 212

Input

1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Cuboid workpiece blank

The NC function includes the following syntax elements:

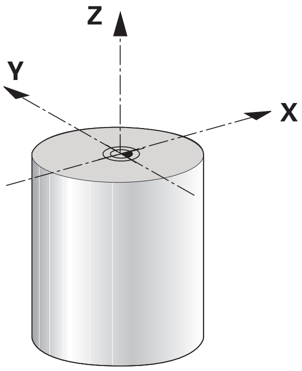
Syntax element	Meaning
BLK FORM	Syntax initiator for cuboid workpiece blank
0.1	Designation of the first NC block
Z	Tool axis Other possibilities might be available, depending on the machine.
X Y Z	Coordinate definition of the MIN point
0.2	Designation of the second NC block
X Y Z	Coordinate definition of the MAX point

9.1.2 Cylindrical workpiece blank with BLK FORM CYLINDER

Application

With **BLK FORM CYLINDER** you define a cylindrical workpiece blank. You can define a cylinder either as a solid piece or as a hollow pipe.

Description of function



Cylindrical blank

To define the cylinder, enter at least the radius or diameter and the height.

The workpiece preset is in the cylinder center in the working plane. Optionally you can define an oversize and the inside radius or diameter of the blank.

Input

1 BLK FORM CYLINDER Z R50 L105 DIST +5 RI10	; Cylindrical blank

The NC function includes the following syntax elements:

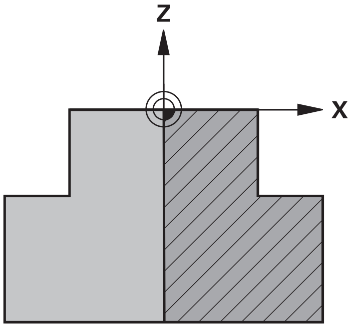
Syntax element	Meaning
BLK FORM CYLINDER	Syntax initiator for cylindrical workpiece blank
Z	Tool axis Other possibilities might be available, depending on the machine.
R or **D**	Radius or diameter of the cylinder
L	Total height of the cylinder
DIST	Oversize of the cylinder relative to the workpiece preset Optional syntax element
RI or **DI**	Inside radius diameter of the core hole Optional syntax element

9.1.3 Rotationally symmetric workpiece blank with BLK FORM ROTATION

Application

With **BLK FORM ROTATION** you define a rotationally symmetric workpiece blank with a definable contour. You define the contour in a subprogram or separate NC program.

Description of function



Blank contour with tool axis **Z** and main axis **X**

In the workpiece blank definition you refer to the contour description.

In the contour description, you program a half-section of the contour around the tool axis as the rotational axis.

The following conditions apply to the contour description:

- Only coordinates of the main axis and tool axis
- Starting point defined in both axes
- Closed contour
- Only positive values in the main axis
- Positive and negative values are possible in the tool axis

The workpiece preset is in the center of the blank in the working plane. You define the coordinates of the blank contour relative to the workpiece preset. You can also define an oversize.

Input

1 BLK FORM ROTATION Z DIM_R LBL "BLANK"	; Rotationally symmetric blank
* - ...	
11 LBL "BLANK"	; Subprogram start
12 L X+0 Z+0	; Beginning of contour
13 L X+50	; Coordinates in positive direction of main axis
14 L Z+50	
15 L X+30	
16 L Z+70	
17 L X+0	
18 L Z+0	; End of contour
19 LBL 0	; End of subprogram

The NC function includes the following syntax elements:

Syntax element	Meaning
BLK FORM ROTATION	Syntax initiator for rotationally symmetric workpiece blank
Z	Active tool axis Other possibilities might be available, depending on the machine.
DIM_R or **DIM_D**	Interpret values in the main axes in the contour description as radius or diameter
LBL or **FILE**	Name or number of the contour subprogram or path of the separate NC program

Notes

- If you program the contour description with incremental values, the control interprets the values as radii regardless of whether **DIM_R** or **DIM_D** is selected.
- With software option 42 CAD Import, you can load contours from CAD files and save them in subprograms or separate NC programs.

  Further information: "Opening CAD Files with the CAD-Viewer", Page 1355

9.1.4 STL file as workpiece blank with BLK FORM FILE

Application

You can integrate 3D models in STL format as workpiece blank and optionally as finished part. This function is particularly convenient in combination with CAM programs, where the required 3D models are available in addition to the NC program.

Requirement

- Max. 20 000 triangles per STL file in ASCII format
- Max. 50 000 triangles per STL file in binary format

Description of function

The dimensions of the NC program come from the same source as the dimensions of the 3D model.

Input

1 BLK FORM FILE "TNC:\CAD\blank.stl" TARGET "TNC:\CAD\finish.stl"	; STL file as workpiece blank and finished part

The NC function includes the following syntax elements:

Syntax element	Meaning
BLK FORM FILE	Syntax initiator for an STL file as workpiece blank
" "	Path of the STL file
TARGET	STL file as finished part Optional syntax element
" "	Path of the STL file

Notes

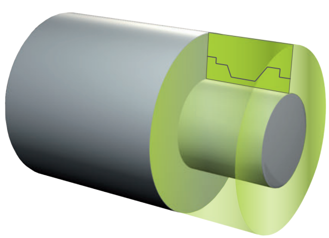
- In the **Simulation** workspace you can export the current view of the workpiece as an STL file. This function allows you to create missing 3-D models, for example semifinished parts if there are several machining steps.
  Further information: "Exporting a simulated workpiece as STL file", Page 1415
- If you have integrated a workpiece blank and a finished part, you can compare the models in the simulation and easily identify any residual material.
  Further information: "Model comparison", Page 1420
- The control loads STL files in binary format faster than STL files in ASCII format.

9.2 Blank form update in turning mode with FUNCTION TURNDATA BLANK (option 50)

Application

Using the blank form update feature, the control detects the already machined areas and adapts all approach and departure paths to the specific, current machining situation. Thus, air cuts are avoided and the machining time is significantly reduced.

You define the workpiece blank for blank form update in a subprogram or separate NC program.



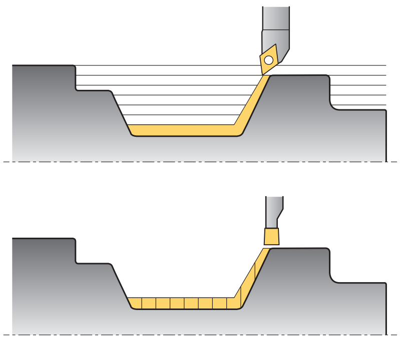
Related topics

- Subprograms
  Further information: "Subprograms and program section repeats with the label LBL", Page 332
- Turning mode: **FUNCTION MODE TURN**
  Further information: "Fundamentals", Page 212
- Defining a workpiece blank with **BLK FORM** for simulation
  Further information: "Defining a workpiece blank with BLK FORM", Page 234

Requirements

- Combined milling/turning (software option 50)
- **FUNCTION MODE TURN** must be active
  Blank form update is only possible with cycle machining in turning mode.
- Closed blank contour for blank form updating
  The starting and end positions must be identical. The workpiece blank corresponds to the cross-section of a rotationally symmetrical body.

Description of function



With **TURNDATA BLANK** you call a contour description used by the control as an updated workpiece blank.

You can define the workpiece blank in a subprogram within the NC program or as a separate NC program.

You can select files or subprograms in the following ways:

- Enter the file path
- Enter the number or name of the subprogram
- Select a file or subprogram by means of a selection window
- Define the file path or name of the subprogram in a QS parameter
- Define the number of the subprogram in a Q, QL, or QR parameter

Use **FUNCTION TURNDATA BLANK OFF** to deactivate blank form update.

Input

1 FUNCTION TURNDATA BLANK LBL "BLANK"	; Blank form update with a workpiece blank from the subprogram "BLANK"
*** - ...**	
11 LBL "BLANK"	; Subprogram start
12 L X+0 Z+0	; Beginning of contour
13 L X+50	; Coordinates in positive direction of main axis
14 L Z+50	
15 L X+30	
16 L Z+70	
17 L X+0	
18 L Z+0	; End of contour
19 LBL 0	; End of subprogram

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION TURNDATA BLANK	Syntax initiator for blank form update in turning mode
OFF, **File**, **QS** or **LBL**	Deactivate blank form update, blank contour as separate NC program, or call as subprogram
Number, **Name** or **QS**	Number or name of the separate NC programor subprogram Fixed or variable number or name If **File**, **QS** or **LBL** is selected

10

Tools

10.1 Fundamentals

To use the control's functions, you must define the tools for the control using real data, e. g. the radius. This makes programming easier and improves process reliability.

To add a tool to the machine, follow the sequence below:

- Prepare your tool and clamp the tool into a suitable tool holder.
- To measure the tool dimensions, starting from the tool carrier preset, measure the tool, e. g. using a tool presetter. The control needs these dimensions for calculating the paths.

  Further information: "Tool carrier reference point", Page 245
- Further tool data are needed to completely define the tool. Take these tool data e. g. from the manufacturer’s tool catalog.

  Further information: "Tool data for the tool types", Page 257
- Save all collected tool data of this tool in the tool management.

  Further information: "Tool management ", Page 270
- As needed, assign a tool carrier to the tool in order to achieve realistic simulation and collision protection.

  Further information: "Tool carrier management", Page 274
- After finishing tool definition, program a tool call within an NC program.

  Further information: "Tool call by TOOL CALL", Page 277
- If your machine is equipped with a chaotic tool changer system and a double gripper, the tool change time may be shortened by pre-selecting the tool.

  Further information: "Tool pre-selection by TOOL DEF", Page 283
- If needed, perform a tool usage test before starting the program. This process checks if the tools are available in the machine and have sufficient remaining tool life.

  Further information: "Tool usage test", Page 284
- After machining a workpiece and measuring it, you may correct the tools.

  Further information: "Tool radius compensation", Page 1038

10.2 Presets on the tool

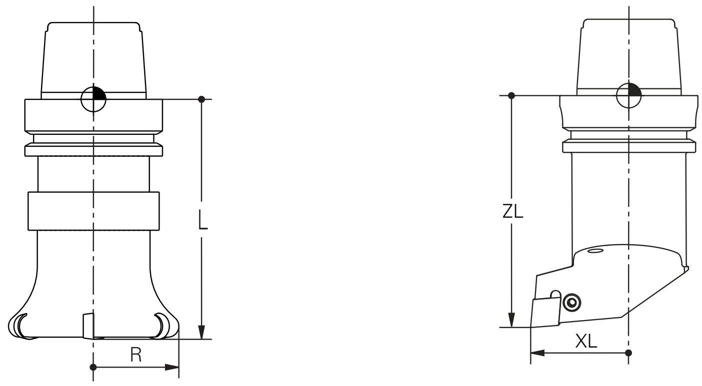
The control distinguishes the following presets on the tool for different calculations or applications.

Related topics

- Presets in the machine or on the workpiece

  Further information: "Presets in the machine", Page 187

10.2.1 Tool carrier reference point

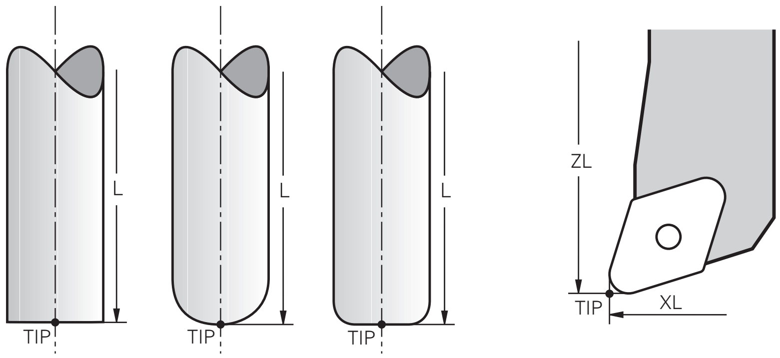


The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

Starting from the tool carrier reference point, define the tool dimensions in the tool management, e. g. length **L** and radius **R**.

Further information: "Tool management ", Page 270

10.2.2 Tool tip TIP



The tool tip has the greatest distance from the tool carrier reference point. The tool tip is the origin of the tool coordinate system **T-CS**.

Further information: "Tool coordinate system T-CS", Page 946

In case of milling cutters, the tool tip is at the center of the tool radius **R** and at the longest point of the tool on the tool axis.

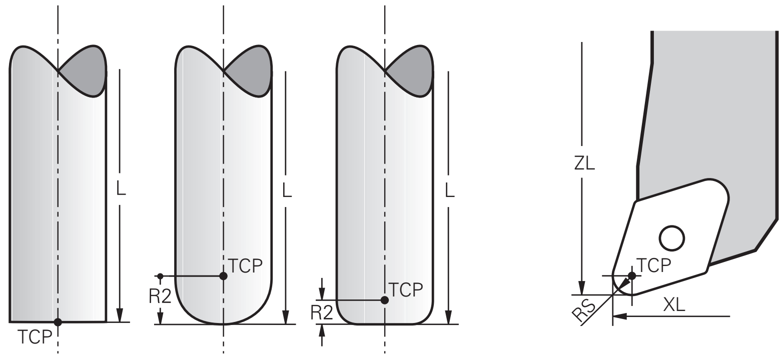
You define the tool tip with the following columns of the tool management relative to the tool carrier reference point:

- **L**
- **DL**
- **ZL** (option 50, option 156)
- **XL** (option 50, option 156)
- **YL** (option 50, option 156)
- **DZL** (option 50, option 156)
- **DXL** (option 50, option 156)
- **DYL** (option 50, option 156)
- **LO** (option 156)
- **DLO** (option 156)

Further information: "Tool data for the tool types", Page 257

In the case of lathe tools (option 50), the control uses the theoretical tool tip, i. e. the point of intersection of the defined values **ZL**, **XL** and **YL**.

10.2.3 Tool center point (TCP, tool center point)



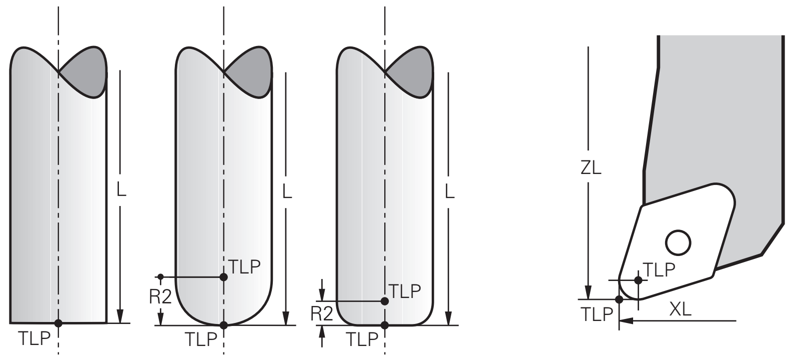
The tool center point is the center of the tool radius **R**. If a second tool radius (**R2**) is defined, the tool center point is offset from the tool tip by this value.

In the case of turning tools (option 50), the tool center point is at the center of cutter radius **RS**.

Making entries in the tool management relative to the tool carrier reference point defines the tool center point.

Further information: "Tool data for the tool types", Page 257

10.2.4 Tool location point (TLP, tool location point)

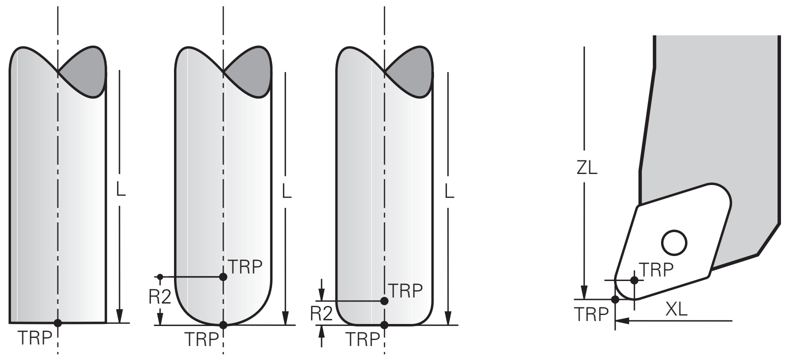


The control positions the tool on the tool location point. By default, the tool location point is at the tool tip.

In the **FUNCTION TCPM** function (option 9), you can also choose the tool location point to be at the tool center point.

Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027

10.2.5 Tool rotation point (TRP, tool rotation point)



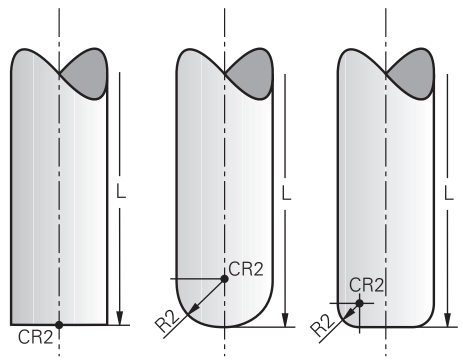
When applying the tilting function with **MOVE** (option 8), the control tilts around the tool rotation point. By default, the tool rotation point is at the tool tip.

When selecting **MOVE** in **PLANE** functions, the syntax element **DIST** is used to define the relative position between the workpiece and the tool. The control shifts the tool rotation point from the tool tip by this value. When **DIST** is not defined, the control keeps the tool tip constant.

Further information: "Rotary axis positioning", Page 1013

In the **FUNCTION TCPM** function (option 9), you can also choose the tool rotation point to be at the tool center point.

10.2.6 Tool radius 2 center (CR2, center R2)



The control uses the tool radius 2 center in conjunction with 3D tool compensation (option 9). In the case of straight lines **LN**, the surface normal vector points to that point and defines the direction of the 3D tool compensation.

Further information: "3D tool compensation (option 9)", Page 1049

The tool radius 2 center is offset from the tool tip and the cutting edge by the **R2** value.

10.3 Tool data

10.3.1 Tool ID number

Application

Each tool has a unique number which equals the line number of the tool management. Each tool ID number is unique.

Further information: "Tool management ", Page 270

Description of function

The tool ID numbers can be defined in a range from 0 to 32,767.

The tool with the number 0 is defined as the zero tool with the length and the radius 0. Upon a TOOL CALL 0, the control unloads the currently used tool and inserts no new tool.

Further information: "Tool call", Page 277

10.3.2 Tool name

Application

A tool name can be assigned in addition to the tool ID number. Contrary to the tool ID number, a tool name is not unique.

Description of function

The tool name allows identifying tools easier within the tool management. To this end, key features can be defined such as the diameter or the type of machining, e. g. **MILL_D10_ROUGH**.

As tool names are not unique, assign names that clearly identify the tools.

A tool name may contain up to 32 characters.

Permitted characters

You can use the following characters for the tool name:

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z 0 1 2 3 4 5 6 7 8 9 # $ % & , - _ .

When entering lowercase letters, the control will substitute them by uppercase letters upon saving.

Note

- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:

  - Tool that is in the spindle
  - Tool that is in the magazine

    > Refer to your machine manual.
    > If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with least remaining tool life.

10.3.3 Indexed tool

Application

Using an indexed tool, several different tool data can be stored for one physically available tool. This feature enables indication of a certain point on the tool by means of the NC program which does not necessarily have to correspond with the maximum tool length.

Description of function

Tools with multiple lengths and radii cannot be defined in one line of the tool management table. Additional table lines are required, specifying the full definitions of the indexed tools. The lengths of the indexed tools approach the tool carrier preset as the index rises, starting from the maximum tool length.

Further information: "Tool carrier reference point", Page 245

Further information: "Creating an indexed tool", Page 251

Examples of an application of indexed tools:

- Step drill

  The tool data of the main tool contain the drill tip, which corresponds to the maximum length. The tool steps are defined are indexed tools. This makes the lengths equal the actual tool dimensions.
- NC center drill

  The main tool is used for defining the theoretical tool tip as the maximum length. This can be used e. g. for centering. The indexed tool defines a point along the tool tooth. This can be used e. g. for deburring.
- Cut-off milling cutter or T-slot milling cutter

  The main tool is used for defining the lower point of the cutting edge, which equals the maximum length. The indexed tool defines the upper point of the cutting edge. When using the indexed tool for cutting-off, the specified workpiece height can be directly programmed.

Creating an indexed tool

To create an indexed tool:

- Select the **Tables** operating mode
- Select **Tool management**

Edit

- Activate **Edit**

> The control enables tool management for editing.

Insert tool

- Select **Insert tool**

> The control opens the **Insert tool** pop-up window.

- Define the tool type
- Define the tool number of the main tool, e. g. **T5**

OK

- Press **OK**

> The control adds table line **5**.

- Define all necessary tool data including the maximum tool length.

  Further information: "Tool data for the tool types", Page 257

Insert tool

- Select **Insert tool**

> The control opens the **Insert tool** pop-up window.

- Define the tool type
- Define the tool number of the indexed tool, e. g. **T5.1**

> The main tool number and an index after the dot define an indexed tool.

OK

- Press **OK**

> The control adds table line **5.1**.

- Define all required tool data

  Further information: "Tool data for the tool types", Page 257

> The control does not adopt any main tool data!
>
> The lengths of the indexed tools approach the tool carrier preset as the index rises, starting from the maximum tool length.
>
> **Further information:** "Tool carrier reference point", Page 245

Notes

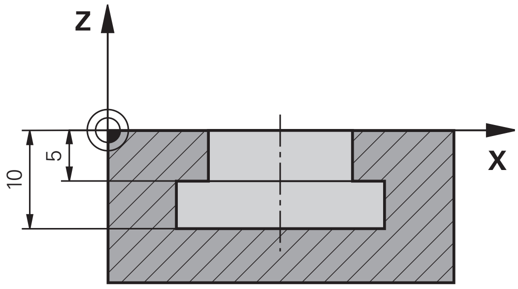
- The control describes some parameters automatically, e. g. the current tool age **CUR_TIME**. The control describes these parameters separately for each table line.

  Further information: "Tool table tool.t", Page 1814
- Index numbers do not need to be sequential. It is possible e. g. to create the tools **T5**, **T5.1** and **T5.3**.
- Up to nine indexed tools can be added to each main tool.

  When defining a replacement tool **RT**, this applies to the respective table line exclusively. When an indexed tool is worn and consequently blocked, this also does not apply to all other indices. This ensures e. g. that the main tool can still be used.

  Further information: "Automatically inserting a replacement tool with M101", Page 1260

Example of T-slot milling cutter



In this example, you program a T-slot with dimensions referring to the top and bottom edges as viewed from the coordinates surface. The height of the T-slot is larger than the length of the cutting edge of the tool used. This requires two steps.

Two tool definitions are required for producing the T-slot.

- The main tool dimension refers to the lower point of the cutting edge, which equals the maximum tool length. This can be used for machining the bottom edge of the T-slot.
- The dimension of the indexed tool refers to the upper point of the cutting edge. This can be used for machining the top edge of the T-slot.

> Please ensure that all required tool data are defined both for the main tool and for the indexed tool! In case of a rectangular tool, the radius remains identical in both table lines.

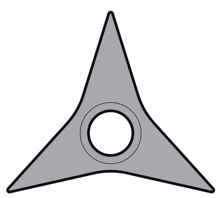
The T-slot is programmed in two machining steps:

- The 10 mm depth is programmed with the main tool.
- The 5 mm depth is programmed with the indexed tool.

11 TOOL CALL 7 Z S2000	; Call the main tool
12 L X+0 Y+0 Z+10 R0 FMAX	; Pre-position the tool
13 L Z-10 R0 F500	; Move to machining depth
14 CALL LBL "CONTOUR"	; Machine the bottom edge of the T-slot with the main tool
*** - ...**	
21 TOOL CALL 7.1 Z F2000	; Call the indexed tool
22 L X+0 Y+0 Z+10 R0 FMAX	; Pre-position the tool
23 L Z-5 R0 F500	; Move to machining depth
24 CALL LBL "CONTOUR"	; Machine the top edge of the T-slot with the indexed tool

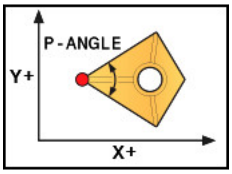
Example FreeTurn tool

You need the following tool data for a FreeTurn tool:



FreeTurn tool with three finishing teeth

> Integrating information about the point angles **P-ANGLE** and the tool length **ZL**, e.g. **FT1_35-35-35_100**, into the tool name is recommended.

Icon and parameter	Meaning	Intended use
ZL	Tool length 1	The tool length **ZL** equals the total tool length, relating to the tool carrier preset. **Further information:** "Presets on the tool", Page 244
XL	Tool length 2	The tool length **XL** equals the difference between the spindle center and the tool tip of the tooth. **XL** must always be defined as a negative value with FreeTrun tools. **Further information:** "Presets on the tool", Page 244
YL	Tool length 3	The tool length **YL** is always 0 with FreeTurn tools.
RS	Cutting radius	You can take the radius **RS** from the tool catalog.
TYPE	Lathe tool type	You select between a rough-turning tool (**ROUGH**) and finishing tool (**FINISH**). **Further information:** "Subgroups of technology-specific tool types", Page 256
TO	Tool orientation	The tool orientation **TO** is always 18 with FreeTurn tools. 
ORI	Angle of orientation	The angle of orientation **ORI** defines the offset of the single teeth with respect to one another. If the first tooth has the value 0, define the second tooth of symmetrical tools at 120 and the third tooth at 240.
P-ANGLE	Point angle	You can get the point angle **P-ANGLE** from the tool catalog.
CUTLENGTH	Cutting-edge length	You can get the tooth length **CUTLENGTH** from the tool catalog.
	Toolcarrier kinematics	Using the optional tool-carrier kinematics, the control can monitor e.g. the tool for collisions Assign the same kinematics to each single tooth.

10.3.4 Tool types

Application

Depending on the selected tool type, the control displays the editable tool data in the tool management.

Related topics

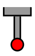
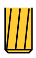
- Editing the tool data in the tool management

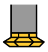
  Further information: "Tool management ", Page 270

Description of function

A number is additionally assigned to each tool type.

The following tool types can be selected in the **TYPE** column of the tool management:

Icon	Tool type	Number
	Milling cutter (**MILL**)	0
	Rough cutter (**MILL_R**)	9
	Finishing cutter (**MILL_F**)	10
	Ball-nose cutter (**BALL**)	22
	Toroid cutter (**TORUS**)	23
	Drill (**DRILL**)	1
	Tap (**TAP**)	2
	NC center drill (**CENT**)	4
	Turning tool (**TURN**) **Further information:** "Types within the turning tools", Page 256	29
	Touch probe (**TCHP**)	21
	Reamer (**REAM**)	3
	Countersink (**CSINK**)	5
	Piloted counterbore (**TSINK**)	6
	Boring tool (**Bor**)	7
	Back boring tool (**BCKBOR**)	8
	Thread miller (**GF**)	1
	Thread miller with chamfer (**GSF**)	16

Icon	Tool type	Number
	Thread mill with single thread (**EP**)	17
	Thread mill with indexable insert (**WSP**)	18
	Thread drilling/milling cutter (**BGF**)	19
	Circular thread mill (**ZBGF**)	20
	Grinding wheel (**GRIND**) **Further information:** "Types within the grinding tools", Page 257	30
	Dressing tool (**DRESS**) **Further information:** "Types within the dressing tools", Page 257	31

These tool types allow filtering the tools in the tool management.

Further information: "Tool management ", Page 270

Subgroups of technology-specific tool types

In the **SUBTYPE** column of the tool management, a technology-specific tool type can be defined, depending on the selected tool type. The control offers the column **SUBTYPE** for the **TURN**, **GRIND** and **DRESS** tool types. Specify the tool type more precisely within these technologies.

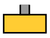
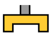
Types within the turning tools

Select between the types below within the turning tools:

Icon	Tool type	Number
	Rough-turning tool (**ROUGH**)	11
	Finish-turning tool (**FINISH**)	12
	Thread-turning tool (**THREAD**)	14
	Recessing tool (**RECESS**)	15
	Button tool (**BUTTON**)	21
	Recess-turning tool (**RECTURN**)	26

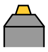
Types within the grinding tools

Select between the types below within the grinding tools:

Icon	Tool type	Number
	Grinding pin (**GRIND_M**)	1
	Special grinding pin (**GRIND_MS**)	2
	Cup wheel (**GRIND_MT**)	3
	Straight wheel (**GRIND_S**) Currently no function	26
	Slant wheel (**GRIND_A**) Currently no function	27
	Facing wheel (**GRIND_P**) Currently no function	28

Types within the dressing tools

Select between the types below within the dressing tools:

Icon	Tool type	Number
	Profile dresser (**DIAMOND**)	101
	Horn-type dresser (**HORNED**) Currently no function	102
	Dressing spindle (**SPINDLE**)	103
	Dressing plate (**PLATE**)	110
	Dressing roll (**ROLL**)	120

10.3.5 Tool data for the tool types

Application

The tool data provide the control with all information necessary for calculating and checking the required movements.

The necessary data depend on the technology and the tool type.

Related topics

- Editing the tool data in the tool management
  Further information: "Tool management ", Page 270
- Tool types
  Further information: "Tool types", Page 255

Description of function

Some of the necessary tool data can be determined using the following options:

- You can measure your tools in the machine (e. g., with a tool touch probe) or externally with a tool presetter.

  Further information: "Touch Probe Cycles: Automatic Tool Measurement", Page 1734
- Take further tool information from the manufacturer's tool catalog, e. g. the material or the number of teeth.

In the tables below, the relevance of the parameters is sub-divided into the optional, recommended and required categories.

The control takes recommended parameters into account for at least one of the functions below:

- Simulation

  Further information: "Simulation of tools", Page 1414
- Machining or touch probe cycles

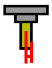
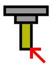
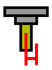
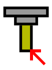
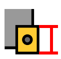
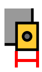
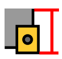
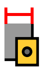
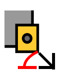
  Further information: "Machining Cycles", Page 419

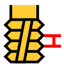
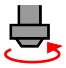
  Further information: "Programmable Touch Probe Cycles", Page 1449
- Dynamic Collision Monitoring (DCM, option 40)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084

Tool data for milling and drilling tools

The control offers the following parameters for milling and drilling tools:

Icon and parameter	Meaning	Intended use
 **L**	Length	Required for all milling and drilling tool types
 **R**	Radius	Required for all milling and drilling tool types
 **R2**	Radius 2	Required for the drilling and milling tool types below: - **Ball-nose cutter** - **Toroid cutter**
 **DL**	Delta value of length	Optional The control describes this parameter in connection with touch probe cycles.
 **DR**	Delta value of radius	Optional The control describes this parameter in connection with touch probe cycles.
 **DR2**	Delta value of radius 2	Optional The control describes this parameter in connection with touch probe cycles.
 **LCUTS**	Tooth length	Recommended
 **RCUTS**	Tooth width	Recommended
 **LU**	Useful length	Recommended
 **RN**	Neck radius	Recommended
 **ANGLE**	Plunge angle	Recommended for the drilling and milling tool types below: - **Milling tool** - **ROUGHING MILL** - **Finishing cutter** - **Ball-nose cutter** - **Toroid cutter**

Icon and parameter	Meaning	Intended use
 PITCH	Thread pitch	Recommended for the drilling and milling tool types below: ■ **Thread mill** ■ **Tapping tools** ■ **Thread miller with chamfer** ■ **Thread mill with single thread** ■ **Thread mill w/ indexable insert** ■ **Thread drilling/milling cutter** ■ **Circular thread mill**
 T-ANGLE	Point angle	Recommended for the drilling and milling tool types below: ■ **Drill** ■ **NC center drill** ■ **Countersink**
 NMAX	Maximum spindle speed	Optional

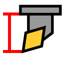
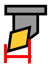
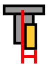
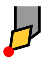
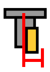
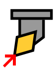
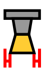
- All tool types listed in the **TYPE** column are milling and drilling tools except for:
  - **Touch probe**
  - **Turning tool**
  - **Grinding wheel**
  - **Dressing tool**

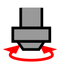
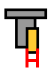
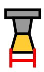
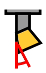
  Further information: "Tool types", Page 255
- The parameters are described in the tool table.

  Further information: "Tool table tool.t", Page 1814

Tool data for turning tools (option 50)

The control offers the following parameters for turning tools:

Icon and parameter	Meaning	Intended use
 **ZL**	Tool length 1	Required for all turning tool types
 **XL**	Tool length 2	Required for all turning tool types
 **YL**	Tool length 3	Required for all turning tool types
 **RS**	Cutting radius	Required for the turning tool types below: ■ **Roughing tool** ■ **Finish-turning tool** ■ **Button tool** ■ **Recessing tool** ■ **Recess-turning tool**
 **TYPE**	Lathe tool type	Required for all turning tool types
 **TO**	Tool orientation	Required for all turning tool types
 **DZL**	Delta value of tool length 1	Optional The control describes this value in connection with touch probe cycles.
 **DXL**	Delta value of tool length 2	Optional The control describes this value in connection with touch probe cycles.
 **DYL**	Delta value of tool length 3	Optional The control describes this value in connection with touch probe cycles.
 **DRS**	Delta value of cutter radius	Optional The control describes this value in connection with touch probe cycles.
 **DCW**	Delta value of cutter width	Optional The control describes this value in connection with touch probe cycles.

Icon and parameter	Meaning	Intended use
 **ORI**	Angle of orientation	Required for all turning tool types
 **T-ANGLE**	Tool angle	Required for the turning tool types below: ■ **Roughing tool** ■ **Finish-turning tool** ■ **Button tool** ■ **Threading tool**
 **P-ANGLE**	Point angle	Required for the turning tool types below: ■ **Roughing tool** ■ **Finish-turning tool** ■ **Button tool** ■ **Threading tool**
  **CUTLENGTH**	Cutting-edge length	Recommended
  **CUTWIDTH**	Tooth width	Required for the turning tool types below: ■ **Recessing tool** ■ **Recess-turning tool** Recommended for the other turning tool types
 **SPB-INSERT**	Angular offset	Required for all turning tool types

- The **TYPE** column of the **Turning tool** tool type as well as the associated technology-specific tool types in the **TYPE** column define turning tools.

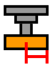
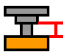
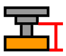
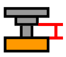
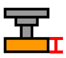
  Further information: "Tool types", Page 255

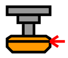
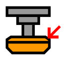
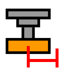
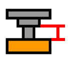
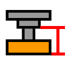
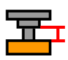
  Further information: "Types within the turning tools", Page 256
- The parameters are described in the turning tool table.

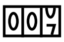
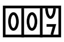
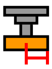
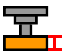
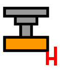
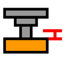
  Further information: "Turning tool table toolturn.trn (option 50)", Page 1823

Tool data for grinding tools (option 156)

The control offers the following parameters for grinding tools:

Icon and parameter	Meaning	Intended use
 **TYPE**	Grinding tool type	Required for all grinding tool types
 **R-OVR**	Radius	Required for all grinding tool types After initial dressing, this value can only be read.
 **L-OVR**	Overhang	Required for the grinding tool types below: ■ **Special grinding pin** ■ **Cup wheel** After initial dressing, this value can only be read.
 **LO**	Overall length	Required for the grinding tool types below: ■ **Grinding pin** ■ **Special grinding pin** After initial dressing, this value can only be read.
 **LI**	Length to the inner edge	Required for the **Special grinding pin** grinding tool type After initial dressing, this value can only be read.
 **B**	Width	Required for the grinding tool types below: ■ **Grinding pin** ■ **Cup wheel** After initial dressing, this value can only be read.
 **G**	Depth of grinding tool	Required for the **Cup wheel** grinding tool type After initial dressing, this value can only be read.
ALPHA	Slant angle	Required for the **Special grinding pin** grinding tool type Unchangeable default value for the grinding tool types below: ■ **Grinding pin** 0° ■ **Cup wheel** 90°
GAMMA	Corner angle	Required for the grinding tool types below: ■ **Special grinding pin** ■ **Cup wheel** Unchangeable default value for the **Grinding pin** 90° tool type

Icon and parameter	Meaning	Intended use
 **RV**	Radius at the edge for **L-OVR**	Optional for the grinding tool types below: ■ **Grinding pin** ■ **Special grinding pin**
 **RV1**	Radius at the edge for **LO**	Optional for the grinding tool types below: ■ **Grinding pin** ■ **Special grinding pin**
 **RV2**	Radius at the edge for **LI**	Optional for the **Special grinding pin** grinding tool type
HW	Wheel has a relief cut	Required for the **Cup wheel** grinding tool type Optional for the remaining grinding tool types
 **HWI**	Angle for a relief cut on the inner edge	Required for the **Cup wheel** grinding tool type Optional for the remaining grinding tool types
 **HWA**	Angle for a relief cut on the outer edge	Required for the **Cup wheel** grinding tool type Optional for the remaining grinding tool types
INIT_D_OK	Initial dressing	Required for all grinding tool types The control activates the checkbox after initial dressing. The checkbox can be cleared, which will require another initial dressing process.
 **dR-OVR**	Delta value of radius	This value can be changed only by cycles.
 **dL-OVR**	Delta value of overhang	This value can be changed only by cycles.
 **dLO**	Delta value of total length	This value can be changed only by cycles.
 **dLI**	Delta value of length up to the inner edge	This value can be changed only by cycles.
 **DRESS-N-D**	Default value of diameter dressing counter	Optional

Icon and parameter	Meaning	Intended use
 **DRESS-N-A**	Default value of outer edge dressing counter	Optional
 **DRESS-N-I**	Default value of inner edge dressing counter	Optional
 **DRESS-N-D-ACT**	Diameter dressing counter	The control increments this value.
 **DRESS-N-A-ACT**	Outer edge dressing counter	The control increments this value.
 **DRESS-N-I-ACT**	Inner edge dressing counter	The control increments this value.
 **R_SHAFT**	Radius of the tool shank	Optional
 **R_MIN**	Min. permissible radius	Optional
 **B_MIN**	Min. permissible width	Optional
 **V_MAX**	Maximum permissible cutting speed	Optional
 **AD**	Retraction amount at the diameter	Required for all grinding tool types
 **AA**	Retraction amount at the outer edge	Required for all grinding tool types
 **AI**	Retraction amount at the inner edge	Required for all grinding tool types

- The **TYPE** column of the **Grinding wheel** tool type as well as the associated technology-specific tool types in the **TYPE** column define grinding tools.

  Further information: "Tool types", Page 255

  Further information: "Types within the grinding tools", Page 257
- The parameters are described in the grinding tool table.

  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 1828

Tool data for dressing tools (option 156)

The control offers the following parameters for dressing tools:

Icon and parameter	Meaning	Intended use
ZL	Tool length 1	Required for dressing tool types
XL	Tool length 2	Required for all dressing tool types
YL	Tool length 3	Required for all dressing tool types
RS	Cutting radius	Required for the dressing tool types below: ■ **Profile dresser** ■ **Dressing spindle**
CUTWIDTH	Width of tooth	Required for the dressing tool types below: ■ **Dressing plate** ■ **Dressing roll**
TYPE	Dressing tool type	Required for all dressing tool types
TO	Tool orientation	Required for all dressing tool types
DZL	Delta value of tool length 1	Optional
DXL	Delta value of tool length 2	Optional
DYL	Delta value of tool length 3	Optional
DRS	Delta value of cutter radius	Optional
N-DRESS	Tool speed	Required for the dressing tool types below: ■ **Dressing spindle** ■ **Dressing roll**

- The **TYPE** column of the **Dressing tool** tool type as well as the associated technology-specific tool types in the **TYPE** column define dressing tools.

  Further information: "Tool types", Page 255

  Further information: "Types within the dressing tools", Page 257
- The parameters are described in the dressing tool table.

  Further information: "Dressing tool table tooldress.drs (option 156)", Page 1836

Tool data for touch probes

The control offers the following parameters for touch probes:

Icon and parameter	Meaning	Intended use
L	Length	Required
R	Radius	Required
TP_NO	Number in the touch probe table	Required
TYPE	Type of touch probe	Required
F	Probing feed rate	Required
FMAX	Rapid traverse in probing cycle	Optional
F_PREPOS	Pre-positioning at rapid traverse	Required
TRACK	Orienting the touch probe in each probing process	Required
REACTION	Trigger **NCSTOP** or **EMERGSTOP** in case of collision	Required
SET_UP	Set-up clearance	Recommended
DIST	Maximum measuring range	Recommended
CAL_OF1	Center offset in the main axis	Required if **TRACK** = **ON** The control describes this value in connection with the calibration cycle.
CAL_OF2	Center offset in the secondary axis	Required if **TRACK** = **ON** The control describes this value in connection with the calibration cycle.

Icon and parameter	Meaning	Intended use
 **CAL_ANG**	Spindle angle during calibration	Required if **TRACK** = **OFF**

- The **TYPE** column of the **Touch probe** tool type as well as the touch probe model in the **TYPE** column define touch probes.
  Further information: "Tool types", Page 255
- The parameters are described in the touch probe table.
  Further information: "Touch probe table tchprobe.tp", Page 1839

10.4 Tool management

Application

The control displays the tool definitions of all technologies as well as the tools currently present in the tool magazine in the **Tool management** application of the **Tables** operating mode.

The tool management allows adding tools, editing tool data and deleting tools.

Related topics

- Creating new tools
  Further information: "Configuring a tool", Page 130
- Table workspace
  Further information: "Table workspace", Page 1803
- Form workspace
  Further information: "Form workspace for tables", Page 1807

Description of function

You can define up to 32,767 tools in the tool management; this is the maximum number of available table lines.

The control displays all tool data of the tool tables below in the tool management:

- Tool table **tool.t**
  Further information: "Tool table tool.t", Page 1814
- Turning tool table **toolturn.trn** (option 50)
  Further information: "Turning tool table toolturn.trn (option 50)", Page 1823
- Grinding tool table **toolgrind.grd** (option 156)
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 1828
- Dressing tool table **tooldress.drs** (option 156)
  Further information: "Dressing tool table tooldress.drs (option 156)", Page 1836
- Touch probe table **tchprobe.tp**
  Further information: "Touch probe table tchprobe.tp", Page 1839

The control additionally displays the pockets occupied in the magazine from pocket table **tool_p.tch** in the tool management.

Further information: "Pocket table tool_p.tch", Page 1843

Tool data can be edited in the **Table** workspace or in the **Form** workspace. In the **Form** workspace the control shows the correct tool data for each tool type.

Further information: "Tool data", Page 249

Notes

- When creating a new tool, the length **L** and radius **R** columns are empty at first. The control will not insert a tool whose length and radius are missing and will display an error message.
- Tool data of tools still stored in the pocket table cannot be deleted. The tools must be removed from the magazine first.
- When editing tool data, bear in mind that the current tool may have been entered in column **RT** as a replacement tool of another tool!
- If the cursor is within the **Table** workspace and the **Edit** switch is deactivated, a search using the keyboard can be started. The control opens a separate window with an input field and automatically searches for the entered string. If it finds a tool with the entered characters, the control selects this tool. If it finds several tools with this string of characters, you can scroll up and down in the window

10.4.1 Importing and exporting tool data

Application

The control can import and export tool data. This avoids manual editing efforts and possible typing errors. Importing tool data is particularly useful in connection with a tool presetter. Exported tool data can be used e. g. for the tool database of your CAM system.

Description of function

The control transmits tool data as a CSV file.

Further information: "File types", Page 1071

The tool data transfer file is structured as follows:

- The first line contains the tool table column names that are transferred.
- The other lines contain the tool data to be transferred. The order of the data must match the order of the column names in line 1. Decimal numbers are separated by a point.

The column names and the tool data stand between double quotation marks and are separated by semicolons.

Please note the following regarding the transfer file:

- The tool number must be present.
- Any tool data can be imported. The data record does not need to contain all tool table column names or all tool data.
- Missing tool data contain no value between the quotation marks.
- The column names can be arranged in any order. The order of tool data must match the order of column names.

Importing tool data

To import tool data:

- ▶ Select the **Tables** operating mode
- ▶ Select **Tool management**

- ▶ Activate **Edit**
- > The control enables tool management for editing.

- ▶ Select **Import**
- > The control opens a selection window.
- ▶ Select the desired file

- ▶ Select **Import**
- > The control adds the tool data to the tool management.
- > If required, the control opens the **Confirm import** window, e. g. in case of identical tool numbers.
- ▶ Selecting the procedure:
  - **Append**: the control adds the tool data into new lines at the end of the table.
  - **Overwrite**: the control overwrites the initial tool data with the tool data from the transfer file.
  - **Cancel**: the control cancels the import process.

NOTICE

Caution: Data may be lost!

When overwriting existing tool data with the **Overwrite** function, the control will permanently delete the initial tool data!

- ▶ Use this function only with tool data that are no longer needed

Exporting tool data

To export tool data:

- Select the **Tables** operating mode
- Select **Tool management**

Edit

- Activate **Edit**
- > The control enables tool management for editing.
- Mark the tool to be exported
- Open the context menu with a long press or by right-clicking

Further information: "Context menu", Page 1392

- Select **Mark row**
- Mark further tools if required

Export

- Select **Export**
- > The control opens the **Save as** window.
- Select a path

> By default, the control saves the transfer file under **TNC:\table**.

- Enter the file name
- Select the file type

> Select either **TNC7 (*.csv)** or **TNC 640 (*.csv)**. The internal formatting of the transfer files differs. If you wish to use the data in a previous control model, select **TNC 640 (*.csv)**.

Create

- Select **Create**
- > The control will save the file using the selected path.

Notes

NOTICE

Caution: Possible material damage!

If the transfer file contains unknown column names, the control will not accept the data from this column! In this case, the control will perform the operations with an incompletely defined tool.

- Check whether the column names are correct
- After importing, check the tool data and adapt them if required.

- The transfer file must be saved under **TNC:\table**.
- The internal formatting of the transfer files differs:
  - **TNC7 (*.csv)** writes the values between double quotation marks and separates the values by semicolons
  - **TNC 640 (*.csv)** writes the values partly between brackets and separates the values by commas

  The TNC7 can both import and export both transfer files.

10.5 Tool carrier management

Application

Tool carrier management allows parameterizing and assigning tool carriers.

The control represents the tool carriers graphically in the simulation and takes the tool carriers into account by calculation, e. g. in Dynamic Collision Monitoring (DCM, option 40).

Related topics

- **Simulation** workspace
  Further information: "Simulation Workspace", Page 1405
- Dynamic Collision Monitoring (DCM, option 40)
  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084

Description of function

To ensure that the control takes the tool carriers into account in its calculations and in the display:

- Save the tool carrier or tool carrier templates
- Parameterize the tool carrier templates
  Further information: "Parameterizing tool carrier templates", Page 276
- Assign a tool carrier
  Further information: "Assigning a tool carrier", Page 276

> If you are using M3D or STL files instead of tool carrier templates, you can assign the files directly to the tools. The parameterization step is superfluous here.
>
> Tool carriers in STL format must meet the following requirements:
>
> - Max. 20 000 triangles
> - Triangular mesh forms a closed shell
>
> If an STL file does not meet the requirements of the control, then the control issues an error message.
>
> For tool carriers, the same requirements with respect to STL and M3D files apply as for fixtures.
>
> **Further information:** "Options for fixture files", Page 1092

Tool-carrier templates

Many tool carriers only differ from others in terms of their dimensions, but their geometric shape is identical. HEIDENHAIN provides ready-to-use tool carrier templates for downloading. Tool carrier templates are 3D models with fixed geometries but changeable dimensions.

Tool carrier templates must be stored as files with a ***.cft** file name extension in the **TNC:\system\Toolkinematics** directory.

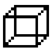
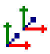
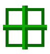
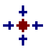
> They can be downloaded through the following link:
>
> **http://www.klartext-portal.com/nc-solutions/en**
>
> If you need further tool carrier templates, please contact your machine manufacturer or third-party vendor.

The tool carrier templates can be parametrized in the **ToolHolderWizard** window. This defines the tool carrier dimensions.

Further information: "Parameterizing tool carrier templates", Page 276

The parametrized tool carriers with the ***.cfx** extension are saved at **TNC:\system \Toolkinematics**.

The **ToolHolderWizard** window contains the following icons:

Icon	Function
	Close the application
	Open file
	Switch between wire frame model and solid object view
	Switch between shaded and transparent view
	Display or hide transformation vectors
	Show or hide names of collision objects
	Display or hide test points
	Show or hide measurement points
	Restore initial view
	Select alignment

10.5.1 Parameterizing tool carrier templates

To parameterize a tool carrier template:

- Select the **Files** operating mode
- Open the **TNC:\system\Toolkinematics** folder
- Double-tap or -click desired tool carrier template with the ***.cft** extension
- > The control opens the **ToolHolderWizard** window.
- Define the dimensions in the **Parameter** area
- Define a name with the ***.cfx** extension in the **Output file** area
- Select **Generate file**
- > The control shows the message that the tool carrier template was successfully generated and saves the file in the folder **TNC:\system\Toolkinematics**.
- Select **OK**
- Select **Quit**

10.5.2 Assigning a tool carrier

To assign a tool carrier to a tool:

- Select the **Tables** operating mode
- Select **Tool management**
- Select the tool you want to use

Edit

- Activate **Edit**
- In the **Spec. functions** panel, select the **KINEMATIC** parameter
- > The control displays the available tool carriers in the **Tool-carrier kinematics** window.
- Select the desired tool carrier

OK

- Select **OK**
- > The control assigns the tool carrier to the tool.

> - The tool carrier will only be taken into account after the next tool call.
> - Parameterized tool carriers can consist of several subfiles. If the subfiles are incomplete, the control will display an error message.
>
>   Only use fully parameterized tool carriers and error-free STL or M3D files!
>
>   For tool carriers, the same requirements with respect to STL and M3D files apply as for fixtures.
>
>   **Further information:** "Fixture monitoring (option 40)", Page 1091

Notes

- In the simulation, the tool carriers can be checked for collisions with the workpiece.

  Further information: "Advanced checks in the simulation", Page 1109
- On 3-axis machines with rectangular angle heads, tool carriers of angle heads are advantageous in connection with the tool axes **X** and **Y** because the control takes the dimensions of the angle heads into account.

  HEIDENHAIN recommends machining with tool axis **Z**. Using Advanced Functions Set 1 (software option 8), you can tilt the working plane to the angle of exchangeable angle heads and continue working with tool axis **Z**.
- The control monitors the tool carriers by means of Dynamic Collision Monitoring (DCM, option 40). This enables the tool carriers to protect against collisions with fixtures or machine components.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084
- A grinding tool that is to be dressed must not contain any tool carrier kinematics (option 156).

10.6 Tool call

10.6.1 Tool call by TOOL CALL

Application

The **TOOL CALL** function calls a tool in the NC program. When the tool is in the tool magazine, the control inserts the tool into the spindle. When the tool is not in the magazine, you can insert it by hand.

Related topics

- Automatic tool change with **M101**

  Further information: "Automatically inserting a replacement tool with M101", Page 1260
- Tool table **tool.t**

  Further information: "Tool table tool.t", Page 1814
- Pocket table **tool_p.tch**

  Further information: "Pocket table tool_p.tch", Page 1843

Requirement

- Tool defined

  To call a tool, the tool must be defined in the tool management.

  Further information: "Tool management ", Page 270

Description of function

Upon calling a tool, the control reads the associated row from the tool management. The tool data are visible on the **Tool** tab of the **Status** workspace.

Further information: "Tool tab", Page 163

> HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.
>
> **Further information:** "Overview of miscellaneous functions", Page 1225

Input

11 TOOL CALL 4 .1 Z S10000 F750 DL +0,2 DR+0,2 DR2+0,2	; Call the tool

The NC function includes the following syntax elements:

Syntax element	Meaning
TOOL CALL	Syntax initiator for a tool call
4, **QS4** or **"MILL_D8_ROUGH"**	Tool definition as a fixed or variable number or name Only the tool definition as a number is unique because the tool names of several tools may be identical! Syntax element depending on technology or application Selection by means of a selection window **Further information:** "Technology-dependent differences when calling tools", Page 279
.1	Step index of the tool Optional syntax element **Further information:** "Indexed tool", Page 250
Z	Tool axis By default, tool axis **Z** is used. Other possibilities might be available, depending on the machine. Syntax element depending on technology or application **Further information:** "Technology-dependent differences when calling tools", Page 279
S or **S(VC =)**	Spindle speed or cutting speed Optional syntax element **Further information:** "Spindle speed S", Page 281
F, **FZ** or **FU**	Feed rate Alternative feed specifications: feed per tooth or feed per revolution Optional syntax element **Further information:** "Feed rate F", Page 282
DL	Delta value of tool length Optional syntax element **Further information:** "Tool compensation for tool length and radius", Page 1036
DR	Delta value of the tool radius Optional syntax element **Further information:** "Tool compensation for tool length and radius", Page 1036
DR2	Delta value of the tool radius 2 Optional syntax element **Further information:** "Tool compensation for tool length and radius", Page 1036

Technology-dependent differences when calling tools

Milling cutter tool call

The following tool data of a milling cutter can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Tool axis
- Spindle speed
- Feed rate
- DL
- DR
- DR2

Calling a milling cutter requires the number or the name of the tool, the tool axis and the spindle speed.

Further information: "Tool table tool.t", Page 1814

Tool call of a turning tool (option 50)

The following tool data of a turning tool can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Feed rate

Calling a turning tool requires the number or the name of the tool.

Further information: "Turning tool table toolturn.trn (option 50)", Page 1823

Tool call of a grinding tool(option 156)

The following tool data of a grinding tool can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Tool axis
- Spindle speed
- Feed rate

Calling a grinding tool requires the number or the name of the tool and the tool axis.

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 1828

Tool call of a dressing tool (option 156)

The following tool data of a dressing tool can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Feed rate

Calling a dressing tool requires the number or the name of the tool!

Further information: "Dressing tool table tooldress.drs (option 156)", Page 1836

A dressing tool can be called only in dressing mode!

Further information: "Activating dressing mode with FUNCTION DRESS", Page 228

Tool call of a workpiece touch probe (option 17)

The following tool data of a workpiece touch probe can be defined:

- Fixed or variable number or name of tool
- Step index of the tool
- Tool axis

Calling a workpiece touch probe requires the number or the name of the tool and the tool axis!

Further information: "Touch probe table tchprobe.tp", Page 1839

Updating tool data

A **TOOL CALL** allows updating the data of the active tool even without tool change, e. g. change the cutting data or delta values. The tool data that can be changed depend on the technology.

In the cases below, the control updates only the data of the active tool:

- Without tool number or tool name and without tool axis
- Without tool number or tool name and with the same tool axis as in the previous tool call

> When a tool number or a tool name or a changed tool axis is programmed in the **TOOL CALL** data record, the control runs a tool change macro.
>
> This may cause the control to insert a replacement tool because the service life has expired.
>
> **Further information:** "Automatically inserting a replacement tool with M101", Page 1260

Notes

- The machine manufacturer uses the machine parameter **allowToolDefCall** (no. 118705) to specify whether a tool can be defined by its name, its number or both in the **TOOL CALL** and **TOOL DEF** functions.

  Further information: "Tool pre-selection by TOOL DEF", Page 283
- The machine manufacturer uses the optional machine parameter **prog-ToolCallDL** (no. 124501) to define whether the control will consider delta values from a tool call in the **Positions** workspace.

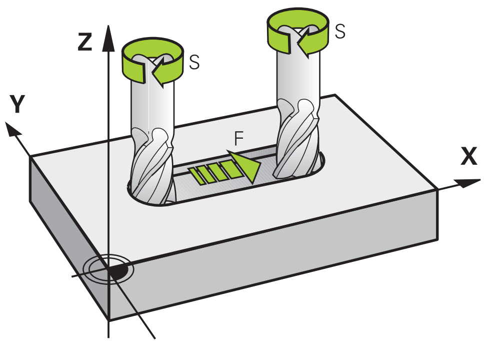
  Further information: "Tool compensation for tool length and radius", Page 1036

  Further information: "Positions workspace", Page 141

10.6.2 Cutting data

Application

The cutting data consist of spindle speed **S** or alternatively constant cutting speed **VC** and feed rate **F**.



Description of function

Spindle speed S

The spindle speed **S** can be defined:

- Tool call by **TOOL CALL**
  Further information: "Tool call by TOOL CALL", Page 277
- Button **S** of **Manual operation** application
  Further information: "Manual operation application", Page 180

The spindle speed **S** is defined in the unit of spindle revolutions per minute (rpm).

Alternatively, the constant cutting speed **VC** in meters per minute (m/min) can be defined.

Further information: "Technology values for turning operations", Page 214

Effect

The spindle speed or the cutting speed is effective until a new spindle speed or cutting speed is defined in a **TOOL CALL** data block.

Potentiometer

The speed potentiometer allows varying the spindle speed between 0 % and 150 % while the program is running. The speed potentiometer setting is effective only for machines with infinitely variable spindle drive. The maximum spindle speed depends on the machine.

Further information: "Potentiometers", Page 101

Status displays

The control displays the current spindle speed in the following workspaces:

- **Positions** workspace
  Further information: "Positions workspace", Page 141
- **POS** tab of the **Status** workspace
  Further information: "POS tab", Page 157

Feed rate F

The feed rate **F** can be defined in the following ways:

- Tool call by **TOOL CALL**

  Further information: "Tool call by TOOL CALL", Page 277
- Positioning block

  Further information: "Path Functions", Page 289
- Button **F** of the **Manual operation** application

  Further information: "Manual operation application", Page 180

The linear axes feed rate is defined in millimeters per minute (mm/min).

The rotary axes feed rate is defined in degrees per minute (°/min).

The feed rate can be defined with an accuracy of three decimal places.

Alternatively, the feed rate can be defined in the NC program or in a tool call in the following units:

- Feed rate per tooth **FZ** in mm/tooth

  FZ defines the path in millimeters that the tool covers per tooth.

  > When using **FZ**, the number of teeth must be defined in the **CUT** column of the tool management.
  >
  > **Further information:** "Tool management ", Page 270

- Feed rate per revolution **FU** in mm/rev

  FU defines the path in millimeters that the tool covers per spindle revolution.

  The feed rate per revolution is used above all when turning (option 50).

  Further information: "Feed rate", Page 216

The feed rate defined in a **TOOL CALL** can be called up within the NC program, using **F AUTO**.

Further information: "F AUTO", Page 282

The feed rate defined in the NC program is effective up to the NC block in which a new feed rate is programmed.

F MAX

When defining **F MAX**, the control moves at rapid traverse. **F MAX** is effective only in the block where it is called. Starting with the following NC block, the last defined feed rate is effective. The maximum feed rate depends on the machine and may depend on the axis.

Further information: "Feed rate limit F MAX", Page 1781

F AUTO

When defining a feed rate in a **TOOL CALL** block, this feed rate can be used in the following positioning blocks, using **F AUTO**.

Button F of the Manual operation application

- If you enter F=0, then the feed rate that the machine manufacturer has defined as minimum feed rate is effective
- If the feed rate entered exceeds the maximum value that has been defined by the machine manufacturer, then the value defined by the machine manufacturer is effective

  Further information: "Manual operation application", Page 180

Potentiometer

The feed rate potentiometer allows varying the feed rate between 0 % and 150 % while the program is running. The setting of the feed rate potentiometer is effective only for the programmed feed rate. As long as the programmed feed rate has not yet been reached, the feed rate potentiometer has no effect.

Further information: "Potentiometers", Page 101

Status displays

The control displays the current feed rate in mm/min in the following workspaces:

- **Positions** workspace

  Further information: "Positions workspace", Page 141
- **POS** tab of the **Status** workspace

  > In the **Manual operation** application, the control displays the feed rate including the decimal points on the **POS** tab. The control displays the feed rate with a total of six decimal points.

  Further information: "POS tab", Page 157
- The control displays the feed rate.
  - When **3D ROT** is active the machining feed rate is shown if several axes are moved
  - If **3-D ROT** is inactive, the feed rate display remains empty when more than one axis is moved simultaneously

  Further information: "3-D rotation window (option 8)", Page 1022

Notes

- In inch programs, the feed rate must be defined in 1/10 inch/min.
- To move your machine at rapid traverse, you can also program the corresponding numerical value, e.g. **F30000**. Unlike **FMAX**, this rapid traverse remains in effect not only in the individual block but in all blocks until you program a new feed rate.
- When moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.

10.6.3 Tool pre-selection by TOOL DEF

Application

Using **TOOL DEF**, the control prepares a tool in the magazine, thus reducing the tool change time.

> Refer to your machine manual.
>
> The preselection of tools with **TOOL DEF** can vary depending on the individual machine tool.

Description of function

If your machine is equipped with a chaotic tool changer system and a double gripper, you can perform tool pre-selection. To do this, program the **TOOL DEF** function after a **TOOL CALL** data record and select the tool to be used next in the NC program. The control prepares the tool while the program is running.

Input

11 TOOL DEF 2 .1	; Tool pre-selection

The NC function includes the following syntax elements:

Syntax element	Meaning
TOOL DEF	Syntax initiator for tool pre-selection
2, **QS2** or **"MILL_D4_ROUGH"**	Tool definition as a fixed or variable number or name Only the tool definition as a number is unique because the tool names of several tools may be identical!
.1	Step index of the tool **Further information:** "Indexed tool", Page 250 Optional syntax element

This function can be used for all technologies except for dressing tools (option 156).

Application example

11 TOOL CALL 5 Z S2000	; Call the tool
12 TOOL DEF 7	; Pre-select the next tool
* - ...	
21 TOOL CALL 7	; Call the pre-selected tool

10.7 Tool usage test

Application

The tool usage test allows checking the tools used in the NC program before starting the program. The control checks if the tools used are available in the machine magazine and have sufficient remaining tool life. Any missing tools can be stored in the machine or tools can be exchanged due to insufficient remaining tool life before starting the program. This avoids interruptions while the program is running.

Related topics

- Contents of the tool usage file
  Further information: "Tool usage file", Page 1845
- Tool usage test in Batch Process Manager (option 154)
  Further information: "Batch Process Manager (option 154)", Page 1768

Requirement

- A tool usage file is needed for performing a tool usage test
  The machine manufacturer uses the machine parameter **createUsageFile** (no. 118701) to define whether the **Generate tool-usage file** function is enabled.
  Further information: "Tool usage file", Page 1845
- The **Generate tool-usage file** function setting is set to either **once** or **always**
  Further information: "Channel settings", Page 1915
- Use the same tool table for the simulation as for the program run
  Further information: "Simulation Workspace", Page 1405

Description of function

Creating the tool usage file

A tool usage file must be generated for performing the tool usage test.

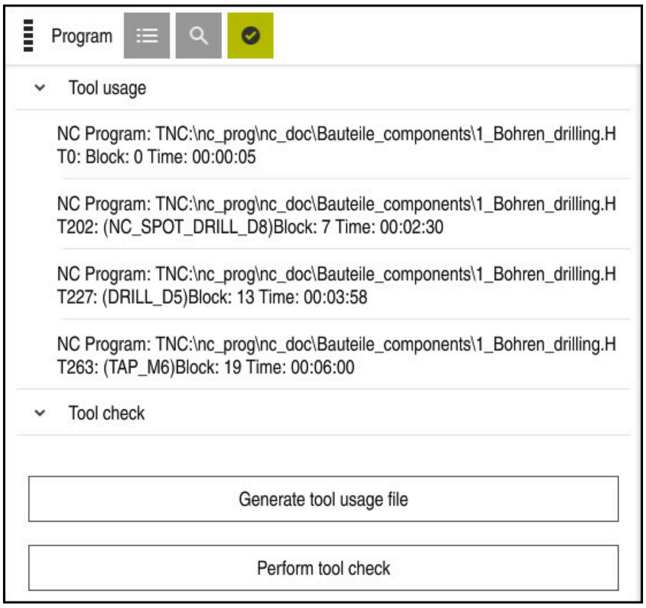
When setting the **Generate tool-usage file** setting to **once** or **always**, the control will generate a tool usage file in the following cases:

- Simulating the NC program completely
- Executing the NC program completely
- Select **Generate tool usage file** in the **Tool check** column of the **Program** workspace

The control saves the tool usage file with the ***.t.dep** extension in the same folder where the NC program is stored.

Further information: "Tool usage file", Page 1845

Tool check column in the Program workspace



Tool check column in the **Program** workspace

The control displays the **Tool usage** and **Tool check** areas in the **Tool check** column of the **Program** workspace.

Further information: "Program workspace", Page 194

Tool usage area

The **Tool usage** area is empty before generating a tool usage file.

Further information: "Creating the tool usage file", Page 285

Further information: "Tool usage file", Page 1845

The control displays the chronological order of all tool calls in the **Tool usage** area, along with the following information:

- Path of NC program in which the tool is called
- Tool number and possibly tool name
- Row number of tool call in NC program
- Tool usage time between the tool changes

Tool check area

Before performing a tool usage test with the **Tool check** button, the **Tool check** area has no content.

Further information: "Performing the tool usage test", Page 287

When performing the tool usage test, the control checks the following:

- The tool is defined in the tool management

  Further information: "Tool management ", Page 270
- The tool is defined in the pocket table

  Further information: "Pocket table tool_p.tch", Page 1843
- The tool has sufficient remaining tool life

  The control checks if the remaining tool life **TIME1** minus **CUR_TIME** is sufficient for the machining process. To meet this requirement, the remaining tool life must be longer than the tool usage time **WTIME** from the tool usage file.

  Further information: "Tool table tool.t", Page 1814

  Further information: "Tool usage file", Page 1845

The control displays the following information in the **Tool check** area:

- **OK**: All tools are available and have sufficient remaining tool life
- **No suitable tool**: The tool is not defined in the tool management

  In this case, check if the correct tool is selected in the tool call. Otherwise, create the tool in the tool management.
- **External tool**: The tool is defined in the tool management, but not in the pocket table

  If your machine is equipped with a magazine, position the missing tool in the magazine.
- **Insufficient remaining tool life**: The tool is blocked or does not have sufficient remaining tool life

  Change the tool or use a replacement tool.

  Further information: "Tool call by TOOL CALL", Page 277

  Further information: "Automatically inserting a replacement tool with M101", Page 1260

10.7.1 Performing the tool usage test

Use the tool usage test as follows:

- Select the **Home** operating mode
- Select the **Settings** application
- Select the **Machine settings** group
- Select the **Machine settings** menu item
- In the **Channel settings** area, select Generate tool usage file **Once** for the simulation.

  Further information: "Channel settings", Page 1915
- Press **Apply**
- Select the **Editor** operating mode
- Select **Add**
- Select the desired NC program
- Select **Open**
- > The control opens the NC program in a new tab.
- Select the **Tool check** column
- > The control opens the **Tool check** column.
- Select **Generate tool usage file**
- > The control generates a tool usage file and displays the tools used in the **Tool usage** area.

  Further information: "Tool usage file", Page 1845
- Select **Perform tool check**
- > The control performs the tool usage test.
- > The **Tool check** area of the control shows whether all tools are available and have sufficient remaining tool life.

Notes

- If you selected **never** in the **Generate tool-usage file** function, the **Generate tool usage file** button in the **Tool check** column is grayed-out.

  Further information: "Channel settings", Page 1915
- The **Simulation settings** window allows selecting when the control generates a tool usage file for the simulation.

  Further information: "Simulation Workspace", Page 1405
- The control creates dependency files (***.dep**); for example, the tool-usage file in order to perform a tool usage test.

  Further information: "Tool usage file", Page 1845
- The control displays the order of tool calls of the NC program that is active in the program run in the **T usage order** table (option 93).

  Further information: "T usage order (option 93)", Page 1847
- The control shows an overview of all tool calls of the NC program that are active in the program run in the **Tooling list** table (option 93).

  Further information: "Tooling list (option 93)", Page 1849
- Function **FN 18: SYSREAD ID975 NR1** allows querying the tool usage test for an NC program.
- Function **FN 18: SYSREAD ID975 NR2 IDX** allows querying the tool usage test for a pallet table. The pallet table row is defined by **IDX**.
- The machine manufacturer uses the machine parameter **autoCheckPrg** (no. 129801) to define whether the control automatically generates a tool usage file upon selecting an NC program.
- The machine manufacturer uses the machine parameter **autoCheckPal** (no. 129802) to define whether the control automatically generates a tool usage file upon selecting a pallet table.
- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control displays dependency files with the *.dep extension in the file manager. Even if the control displays no dependency files, it still generates a tool usage file.

11

Path Functions

11.1 Fundamentals of coordinate definitions

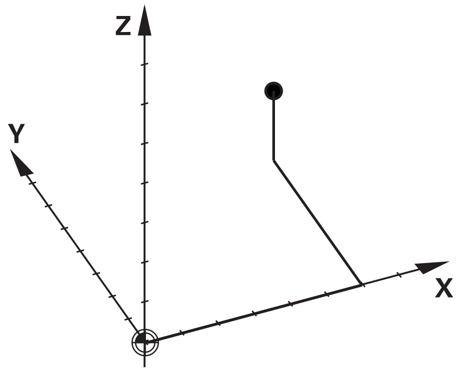
You program a workpiece by defining the path contours and the target coordinates.

Depending on the dimensioning used in the technical drawing, you use Cartesian or polar coordinates with absolute or incremental values.

11.1.1 Cartesian coordinates

Application

A Cartesian coordinate system consists of two or three axes that are all mutually perpendicular. Cartesian coordinates are relative to the datum (origin) of the coordinate system, which is at the intersection of the axes.



With Cartesian coordinates you can uniquely specify a point in space by defining the three axis values.

Description of function

In the NC program you define the values in the linear axes **X**, **Y**, and **Z**, such as with a straight line **L**.

```
11 L  X+60 Y+50 Z+20  RL F200
```

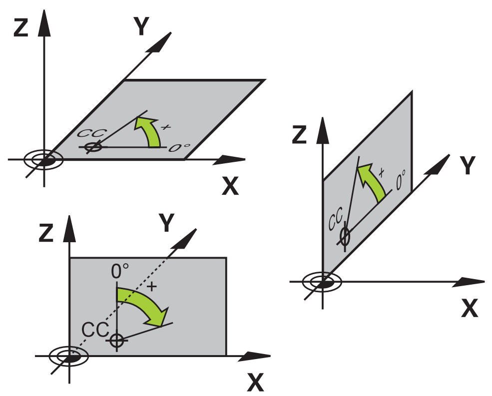
The programmed coordinates are modally effective. As long as the value of an axis remains the same, you do not need to program the value for further path contours.

11.1.2 Polar coordinates

Application

You define polar coordinates in one of the three planes of a Cartesian coordinate system.

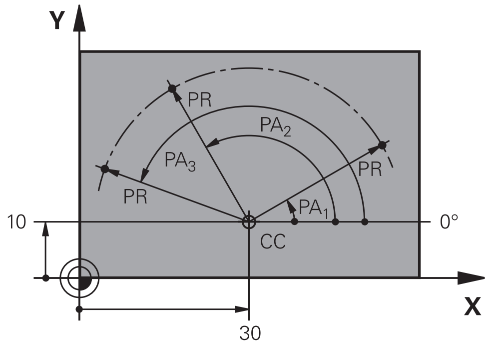
Polar coordinates are relative to a previously defined pole. From this pole you define a point by its distance to the pole and the angle to the angle reference axis.



Description of function

Polar coordinates can be used in, for example, the following situations:

- Points on circular paths
- Workpiece drawings with angular information, such as bolt hole circles



You define the pole **CC** with Cartesian coordinates in two axes. These axes specify the plane and the angle reference axis.

The pole is modally effective within an NC program.

The angle reference axis is related to the plane as follows:

Plane	Angle reference axis
XY	+X
YZ	+Y
ZX	+Z

```
11 CC  X+30 Y+10
```

The polar coordinate radius **PR** is relative to the pole. **PR** defines the distance of this point from the pole.

The polar coordinate angle **PA** defines the angle between the angle reference axis and this point.

```
11 LP  PR+30 PA+10  RR F300
```

The programmed coordinates are modally effective. As long as the value of an axis remains the same, you do not need to program the value for further path contours.

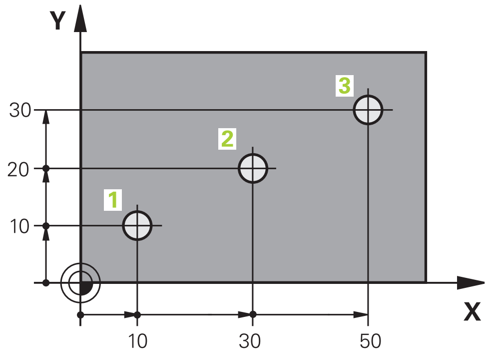
11.1.3 Absolute entries

Application

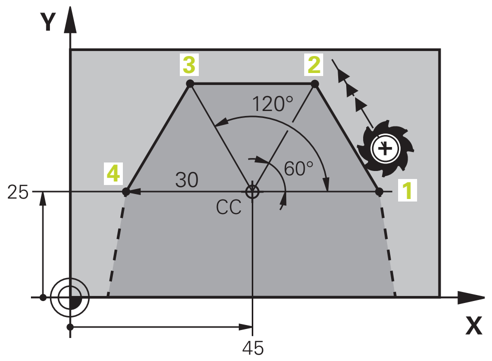
Absolute entries always refer to an origin. For Cartesian coordinates the origin is the datum, and for polar coordinates the origin is the pole and the angle reference axis.

Description of function

Absolute entries define the point that the control positions to.



11 L X+10 Y+10 RL F200 M3	; Position to point 1
12 L X+30 Y+20	; Position to point 2
13 L X+50 Y+30	; Position to point 3



11 CC X+45 Y+25	; Define the pole in two axes with Cartesian coordinates
12 LP PR+30 PA+0 RR F300 M3	; Position to point 1
13 LP PA+60	; Position to point 2
14 LP PA+120	; Position to point 3
15 LP PA+180	; Position to point 4

11.1.4 Incremental entries

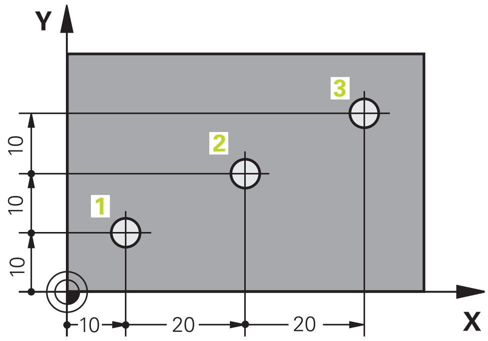
Application

Incremental entries are always referenced to the previously programmed coordinates. For Cartesian coordinates those are the values in the axes **X**, **Y**, and **Z**, and for polar coordinates the value of the polar coordinate radius **PR** and the polar coordinate angle **PA**.

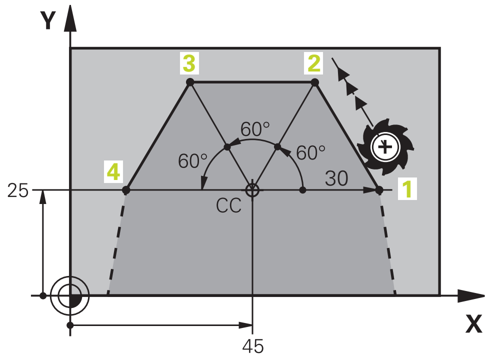
Description of function

Incremental entries define the value by which the control positions. The previously programmed coordinates serve as the respective datum of the coordinate system.

You define incremental coordinates with an **I** before each axis designation.



11 L X+10 Y+10 RL F200 M3	; Position to point 1 absolutely
12 L IX+20 IY+10	; Position to point 2 incrementally
13 L IX+20 IY+10	; Position to point 3 incrementally



11 CC X+45 Y+25	; Define the pole absolutely in two axes with Cartesian coordinates
12 LP PR+30 PA+0 RR F300 M3	; Position to point 1 absolutely
13 LP IPA+60	; Position to point 2 incrementally
14 LP IPA+60	; Position to point 3 incrementally
15 LP IPA+60	; Position to point 4 incrementally

11.2 Fundamentals of path functions

Application

When creating an NC program, you can use the path functions to program the individual contour elements. To do so, use coordinates to define the end points of the contour elements.

The control then uses the coordinate entries, the tool data, and the radius compensation to calculate the traverse path. The control simultaneously positions all machine axes that you programmed in the NC block of a path function.

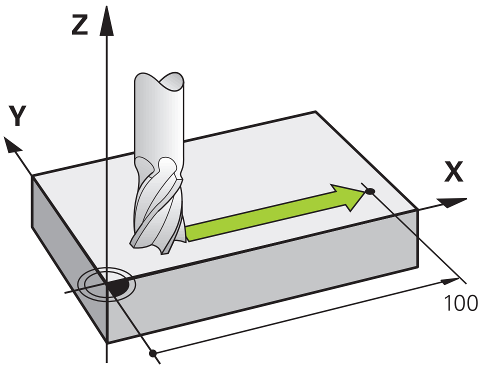
Description of function

Inserting a path function

The gray path function keys initiate the dialog. The control inserts the NC block in the NC program and prompts you for each piece of necessary information.

> Depending on the design of the machine tool, either the tool moves or the machine table moves. When programming a path function, you always assume that the tool is in motion.

Motion in one axis



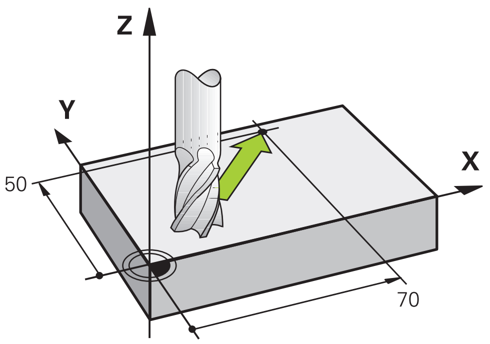
If the NC block contains one coordinate, the control moves the tool parallel to the programmed machine axis.

Example

```
L  X+100
```

The tool retains the Y and Z coordinates and moves to the position **X+100**.

Motion in two axes



If the NC block contains two coordinates, the control moves the tool in the programmed plane.

Example

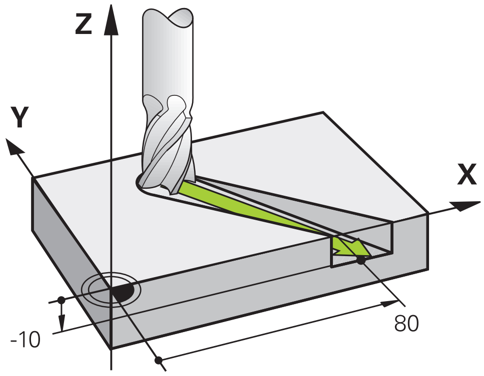
```
L X+70  Y+50
```

The tool retains the Z coordinate and moves in the XY plane to the position **X+70 Y+50**.

You define the working plane by entering the tool axis when calling the tool with **TOOL CALL**.

Further information: "Designation of the axes on milling machines", Page 186

Motion in more than two axes



If the NC block contains three coordinate entries, the control moves the tool spatially to the programmed position.

Example

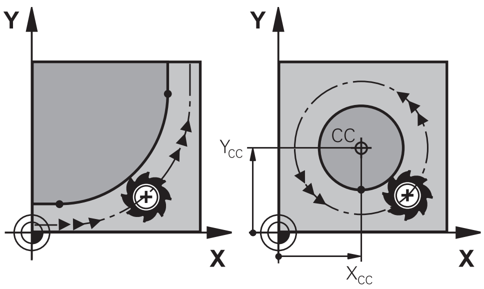
```
L X+80  Y+0 Z-10
```

Depending on the kinematics of your machine, you can program up to six axes in a linear **L** block.

Example

```
L X+80 Y+0 Z-10 A+15 B+0 C-45
```

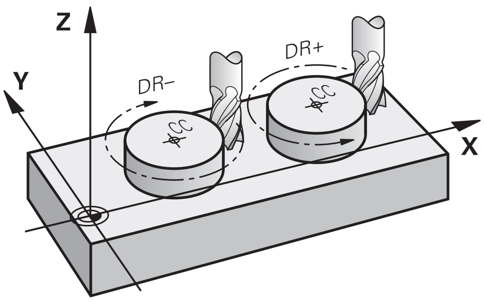
Circles and arcs



Use the path functions for circular arcs to program circular motions in the working plane.

The control moves the tool in two axes simultaneously on a circular path relative to the workpiece. You can program circular paths with a circle center point **CC**.

Direction of rotation DR for circular motions



When a circular path has no tangential transition to another contour element, define the direction of rotation as follows:

- Clockwise direction of rotation: **DR-**
- Counterclockwise direction of rotation: **DR+**

Tool radius compensation

Tool radius compensation is defined in the NC block of the first contour element.

Do not activate tool radius compensation in an NC block for a circular path. Activate tool radius compensation in a preceding straight line.

Further information: "Tool radius compensation", Page 1038

Pre-positioning

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning can also lead to contour damage. There is danger of collision during the approach movement!

- Program a suitable pre-position
- Check the sequence and contour with the aid of the graphic simulation

11.3 Path functions with Cartesian coordinates

11.3.1 Overview of path functions

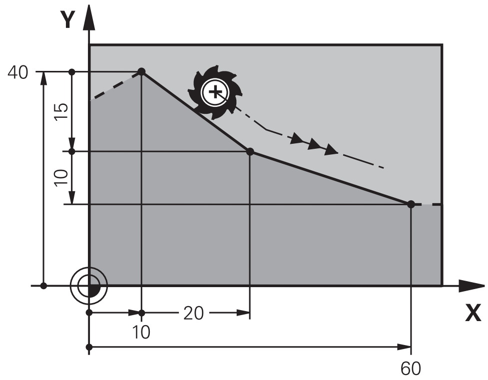
Key	Function	Further information
L	Straight line **L** (line)	Page 297
CHF	Chamfer **CHF** (chamfer) Chamfer between two straight lines	Page 297
RND	Rounding **RND** (rounding of corner) Circular arc with tangential connection to the preceding and subsequent contour elements	Page 299
CC	Circle center point **CC** (circle center)	Page 300
C	Circular path **C** (circle) Circular path around a circle center **CC** to an end point	Page 300
CR	Circular path **CR** (circle by radius) Circular path with a specified radius	Page 302
CT	Circular path **CT** (circle tangential) Circular path with tangential connection to the preceding contour element	Page 303

11.3.2 Straight line L

Application

With a straight line **L** you program a straight traverse motion in any direction.

Description of function



The control moves the tool in a straight line from its current position to the defined end point. The starting point is the end point of the preceding NC block.

Depending on the kinematics of your machine, you can program up to six axes in a linear **L** block.

Programming a straight line L

To program a straight line:

- Select **L**
- Define the coordinates of the end point
- Select radius compensation, if necessary
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary

Example

```
7 L  X+10 Y+40  RL F200  M3
8 L IX+20 IY-15
9 L X+60 IY-10
```

11.3.3 Chamfer CHF

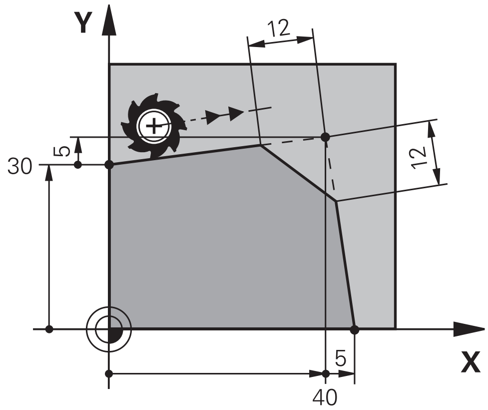
Application

With the chamfer function **CHF** you can insert a chamfer at the intersection of two straight lines.

Requirements

- Straight lines in the working plane before and after the chamfer
- Identical tool compensation before and after the chamfer
- Chamfer is machinable with the current tool

Description of function



Cutting two straight lines creates contour corners. You can insert a chamfer at these contour corners. The angle of the corner is irrelevant; you simply define the length by which each straight line is shortened. The control does not traverse to the corner point.

If you program a feed rate in the **CHF** block, then this feed rate is in effect only while cutting the chamfer.

Programming a chamfer

To program a chamfer:

- Select **CHF**
- Define the length of the chamfer
- Define the feed rate, if necessary

Example

7 L X+0 Y+30 RL F300 M3
8 L X+40 IY+5
9 CHF 12 F250
10 L IX+5 Y+0

11.3.4 Rounding RND

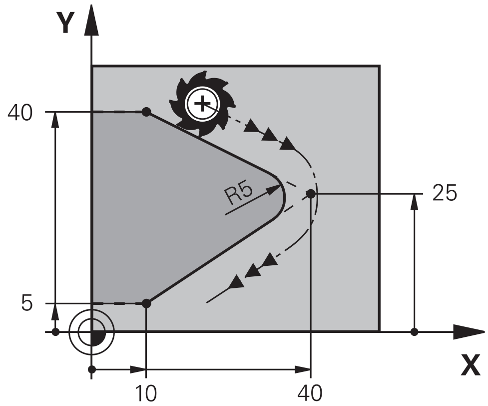
Application

With the rounding function **RND** you can use a circular path to round off the intersection of two path functions.

Requirements

- Path functions before and after the rounding arc
- Identical tool compensation before and after the rounding arc
- Rounding is machinable with the current tool

Description of function



You program the rounding arc between two path functions. The circular arc connects tangentially to the previous and subsequent contour element. The control does not traverse to the intersection.

If you program a feed rate in the **RND** block, then this feed rate is in effect only while cutting the rounding arc.

Program rounding RND

To program a rounding **RND**:

- Select **RND**
- Define the radius
- Define the feed rate, if necessary

Example

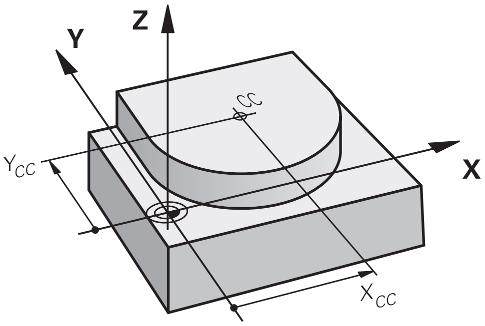
```
5 L  X+10 Y+40  RL F300  M3
6 L  X+40 Y+25
7 RND  R5 F100
8 L  X+10 Y+5
```

11.3.5 Circle center point CC

Application

You use the circle center point function **CC** to define a position as circle center point.

Description of function



You define a circle center point by entering coordinates for at most two axes. If you do not enter coordinates, the control uses the last defined position. The circle center point remains active until you define a new circle center point. The control does not traverse to the circle center point.

You need to define a circle center point before you can program a circular path with **C**.

> The control simultaneously uses the **CC** function as the pole for polar coordinates.
>
> **Further information:** "Polar coordinate datum at pole CC", Page 307

Programming the circle center point CC

To program a circle center point **CC**:

- Select **CC**
- Define the coordinates of the circle center point

Example

```
5 CC  X+25 Y+25
```

or

```
10 L  X+25 Y+25
11 CC
```

11.3.6 Circular path C

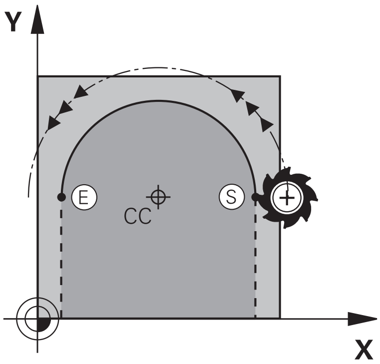
Application

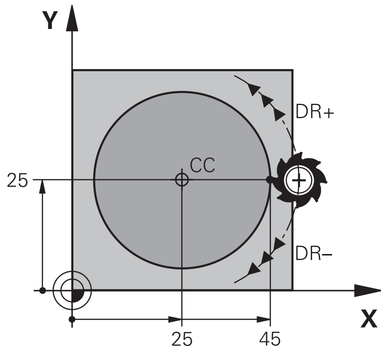
You use the circular path function **C** to program a circular path around a circle center point.

Requirement

- Circle center point **CC** is defined
  Further information: "Circle center point CC", Page 300

Description of function





The control moves the tool on a circular path from the current position to the defined end point. The starting point is the end point of the preceding NC block. You can use at most two axes to define the new end point.

If you want to program a full circle, then define the same coordinates for the starting and end point. These points must lie on the circular path.

> In the machine parameter **circleDeviation** (no. 200901) you can define the permissible deviation of the circle radius. The maximum permissible deviation is 0.016 mm.

With the direction of rotation you define whether the control moves along the circular path in a clockwise or counterclockwise direction.

Definition of the direction of rotation:

- Clockwise: direction of rotation **DR-** (with radius compensation **RL**)
- Counterclockwise: direction of rotation **DR+** (with radius compensation **RL**)

Programming a circular path C

To program a circular path with **C**:

- Select **C**
- Define the coordinates of the end point
- Select the direction of rotation
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary

Example

```
5 CC  X+25 Y+25
6 L  X+45 Y+25  RR F200  M3
7 C  X+45 Y+25  DR+
```

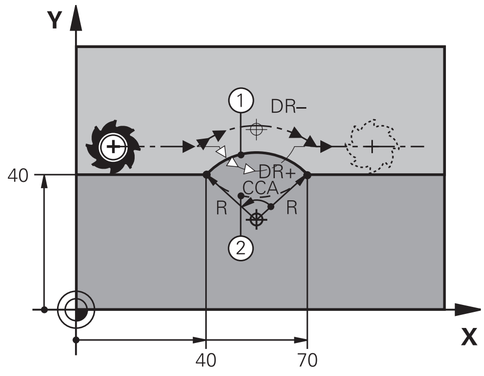
11.3.7 Circular path CR

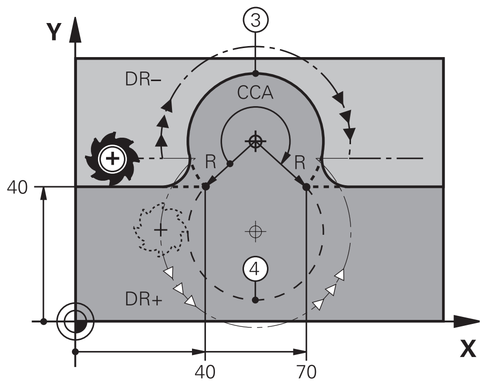
Application

You use a radius to program a circular path with the circular path function **CR**.

Description of function

The control moves the tool on a circular path, with the radius **R**, from the current position to the defined end point. The starting point is the end point of the preceding NC block. You can use at most two axes to define the new end point.





The starting and end points can be connected with four different circular paths of the same radius. You define the correct circular path with the central angle **CCA** of the circular path radius **R** and the direction of rotation **DR**.

The algebraic sign of the circular path radius **R** is decisive for whether the control selects a central angle that is greater than or less than 180°.

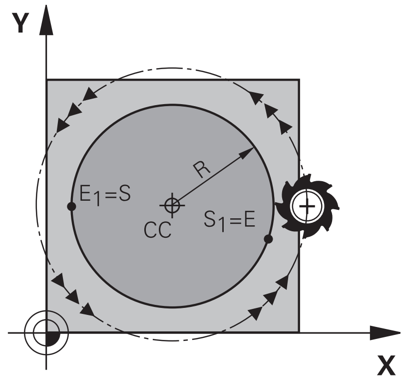
The radius has the following effects on the central angle:

- Smaller circular path: **CCA**<180°
  Radius with a positive sign **R**>0
- Longer circular path: **CCA**>180°
  Radius with a negative sign **R**<0

With the direction of rotation you define whether the control moves along the circular path in a clockwise or counterclockwise direction.

Definition of the direction of rotation:

- Clockwise: direction of rotation **DR-** (with radius compensation **RL**)
- Counterclockwise: direction of rotation **DR+** (with radius compensation **RL**)



For a full circle, program two circular paths in succession. The end point of the first circular path is the starting point of the second. The end point of the second circular path is the starting point of the first.

Programming a circular path CR

To program a circular path with **CR**:

- Select **CR**
- Define the coordinates of the end point
- Define a positive or negative radius
- Select the direction of rotation
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary

Note

The distance between the starting and end points must not be greater than the circle diameter.

Example

10 L X+40 Y+40 RL F200 M3	
11 CR X+70 Y+40 R+20 DR-	; Circular path 1

or

11 CR X+70 Y+40 R+20 DR+	; Circular path 2

or

11 CR X+70 Y+40 R-20 DR-	; Circular path 3

or

11 CR X+70 Y+40 R-20 DR+	; Circular path 4

11.3.8 Circular path CT

Application

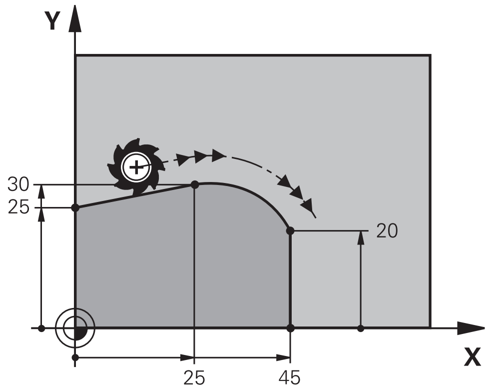
You use the circular path function **CT** to program a circular path that connects tangentially to the previously programmed contour element.

Requirement

- Previous contour element programmed

  Before you can program a circular path with **CT** you must program a contour element to which the circular path can connect tangentially. This requires at least two NC blocks.

Description of function



The control moves the tool on a circular path, with a tangential connection, from the current position to the defined end point. The starting point is the end point of the preceding NC block. You can use at most two axes to define the new end point.

When contour elements uniformly merge into another without kinks, then this transition is referred to as tangential.

Programming a circular path CT

To program a circular path with **CT**:

- Select **CT**
- Define the coordinates of the end point
- Select radius compensation, if necessary
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary

Note

Both coordinates of the contour element must lie in the same plane as the coordinates of the circular path.

Example

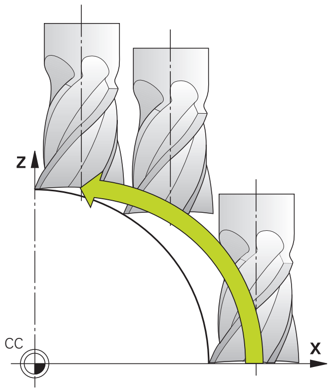
7 L X+0 Y+25 RL F300 M3
8 L X+25 Y+30
9 CT X+45 Y+20
10 L Y+0

11.3.9 Circular path in another plane

Application

You can also program circular paths that do not lie in the active working plane.

Description of function



You program circular paths that lie in another plane by entering one axis of the working plane and the tool axis.

Further information: "Designation of the axes on milling machines", Page 186

You can program circular paths that lie in another plane with the following functions:

- **C**
- **CR**
- **CT**

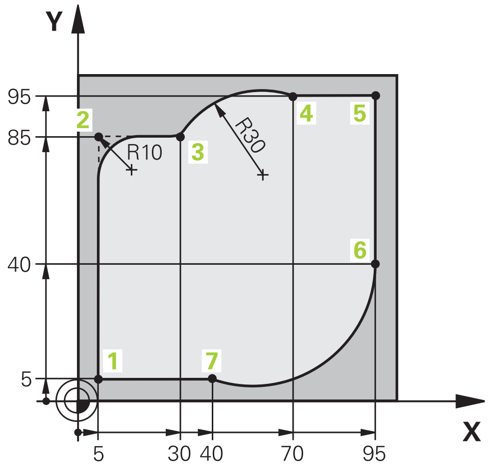
> If you want to use the function **C** for circular paths in another plane, you must first define the circle center point **CC** by entering one of the axes of the working plane and the tool axis.

Spatial arcs are created when these circular paths rotate. When machining spatial arcs, the control moves in three axes.

Example

```
3  TOOL CALL 1 Z S4000
4 ...
5 L  X+45 Y+25 Z+25 RR F200  M3
6 CC  X+25 Z+25
7 C  X+45 Z+25  DR+
```

11.3.10 Example: Cartesian path functions



0 BEGIN PGM CIRCULAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Define the workpiece blank for workpiece simulation
3 TOOL CALL 1 Z S4000	; Call the tool in the tool axis and with the spindle speed
4 L Z+250 R0 FMAX	; Retract the tool in the tool axis at rapid traverse FMAX
5 L X-10 Y-10 R0 FMAX	; Pre-position the tool
6 L Z-5 R0 F1000 M3	; Move to working depth at feed rate F = 1000 mm/min
7 APPR LCT X+5 Y+5 R5 RL F300	; Approach the contour at point 1 on a circular path with tangential connection
8 L X+5 Y+85	; Program the first straight line for corner 2
9 RND R10 F150	; Program a rounding with R = 10 mm, feed rate F = 150 mm/min
10 L X+30 Y+85	; Move to point 3: starting point of the circular path CR
11 CR X+70 Y+95 R+30 DR-	; Move to point 4: end point of the circular path CR, with radius R = 30 mm
12 L X+95	; Move to point 5
13 L X+95 Y+40	; Move to point 6: starting point of the circular path CT
14 CT X+40 Y+5	; Move to point 7: end point of the circular path CT, arc with tangential connection to point 6; the control calculates the radius automatically
15 L X+5	; Move to last contour point 1
16 DEP LCT X-20 Y-20 R5 F1000	; Depart contour on a circular path with tangential connection
17 L Z+250 R0 FMAX M2	; Retract the tool, end program
18 END PGM CIRCULAR MM	

11.4 Path functions with polar coordinates

11.4.1 Overview of polar coordinates

With polar coordinates you can define a position in terms of its angle **PA** and its distance **PR** relative to a previously defined pole **CC**.

Overview of path functions with polar coordinates

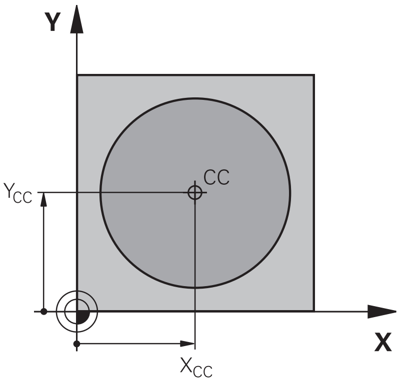
Key	Function	Further information
L + P	Straight line **LP** (line polar)	Page 308
C + P	Circular path **CP** (circle polar) Circular path around circle center point or pole **CC** to arc end point	Page 309
CT + P	Circular path **CTP** (circle tangential polar) Circular path with tangential connection to the preceding contour element	Page 310
C + P	Helix with circular path **CP** (circle polar) Combination of a circular and a linear motion	Page 311

11.4.2 Polar coordinate datum at pole CC

Application

You must define a pole before you can program with polar coordinates. All polar coordinates are relative to the pole.

Description of function



You use the **CC** function to define a position as the pole. You define a pole by entering coordinates for at most two axes. If you do not enter coordinates, the control uses the last defined position. The pole remains active until you define a new pole. The control does not traverse to this position.

Programming the CC pole

To program a **CC** pole:

CC

- Select **CC**
- Define the coordinates of the pole

Example

```
11 CC  X+30 Y+10
```

11.4.3 Straight line LP

Application

With the straight line function **LP** you program a straight traverse motion in any direction using polar coordinates.

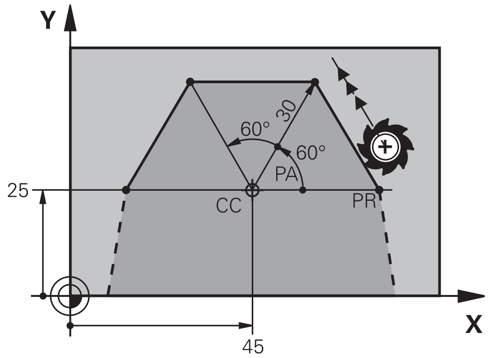
Requirement

- Pole **CC**

  You must define a pole **CC** before programming with polar coordinates.

  Further information: "Polar coordinate datum at pole CC", Page 307

Description of function



The control moves the tool in a straight line from its current position to the defined end point. The starting point is the end point of the preceding NC block.

You define the straight line with the polar coordinate radius **PR** and the polar coordinate angle **PA**. The polar coordinate radius **PR** is the distance from the end point to the pole.

The algebraic sign of **PA** depends on the angle reference axis:

- If the angle from the angle reference axis to **PR** is counterclockwise: **PA**>0
- If the angle from the angle reference axis to **PR** is clockwise: **PA**<0

Programming a straight line LP

To program a straight line:

- Select **L**

- Select **P**
- Define the polar coordinate radius **PR**
- Define the polar coordinate angle **PA**
- Select radius compensation, if necessary
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary

Note

The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate entry.

Further information: "Form column in the Program workspace", Page 201

Example

```
12 CC  X+45 Y+25
13 LP  PR+30 PA+0  RR F300  M3
14 LP  PA+60
15 LP  IPA+60
16 LP  PA+180
```

11.4.4 Circular path CP around pole CC

Application

You use the circular path function **CP** to program a circular path around the defined pole.

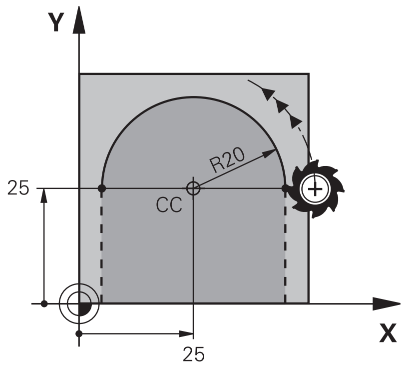
Requirement

- Pole **CC**

  You must define a pole **CC** before programming with polar coordinates.

  Further information: "Polar coordinate datum at pole CC", Page 307

Description of function



The control moves the tool on a circular path from the current position to the defined end point. The starting point is the end point of the preceding NC block.

The distance from the starting point to the pole is automatically both the polar coordinate radius **PR** as well as the radius of the circular path. You define the polar coordinate angle **PA** that the control moves to with this radius.

Programming a circular path CP

To program a circular path with **CP**:

- Select **C**

- Select **P**
- Define the polar coordinate angle **PA**
- Select the direction of rotation
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary

Notes

- In the **Form** workspace you can toggle between the syntaxes for Cartesian and polar coordinate entry.
- If you define **PA** incrementally, you must define the direction of rotation with the same algebraic sign.

  Consider this behavior when importing NC programs from earlier controls, and adapt the NC programs if necessary.

Example

```
18 LP  PR+20 PA+0  RR F250  M3
19 CC  X+25 Y+25
20 CP  PA+180 DR+
```

11.4.5 Circular path CTP

Application

You use the **CTP** function to program a circular path with polar coordinates that connects tangentially to the previously programmed contour element.

Requirements

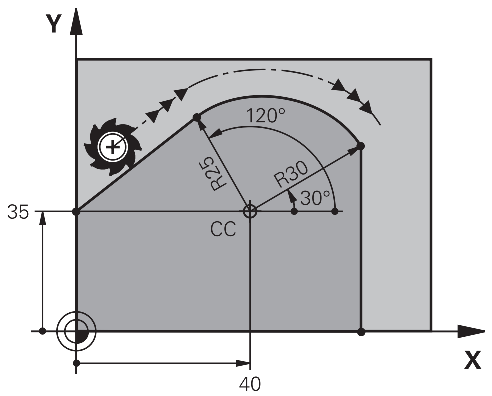
- Pole **CC**

  You must define a pole **CC** before programming with polar coordinates.

  Further information: "Polar coordinate datum at pole CC", Page 307
- Previous contour element programmed

  Before you can program a circular path with **CTP** you must program a contour element to which the circular path can connect tangentially. This requires at least two positioning blocks.

Description of function



The control moves the tool on a circular path, with a tangential connection, from the current position to the end point defined with polar coordinates. The starting point is the end point of the preceding NC block.

When contour elements uniformly merge into another, without kinks or corners, then this transition is referred to as tangential.

Programming a circular path CTP

To program a circular path with **CTP**:

- Select **CT**
- Select **P**
- Define the polar coordinate radius **PR**
- Define the polar coordinate angle **PA**
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary

Notes

- The pole is **not** the center of the contour circle!
- The **Form** column allows toggling between the syntaxes for Cartesian and polar coordinate entry.

  Further information: "Form column in the Program workspace", Page 201

Example

```
12 L  X+0 Y+35  RL F250  M3
13 CC  X+40 Y+35
14 LP  PR+25 PA+120
15 CTP  PR+30 PA+30
16 L  Y+0
```

11.4.6 Helix

Application

A helix is a cylindrical spiral that is programmed on a circular path with a constant pitch.

Requirements

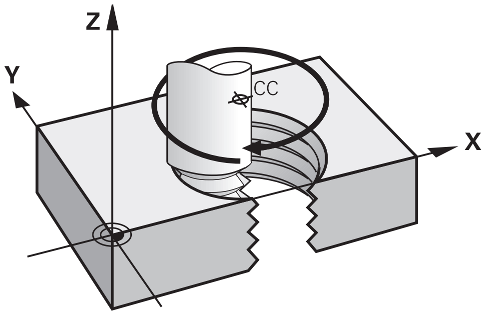
The path contours for a helix can only be programmed with a circular path **CP**.

Further information: "Circular path CP around pole CC", Page 309

You must define a pole **CC** before programming with polar coordinates.

Further information: "Polar coordinate datum at pole CC", Page 307

Description of function



A helix is a combination of a circular path **CP** and a linear motion perpendicular to this path. You program the circular path **CP** in the working plane.

Helices are used in the following cases:

- Large-diameter internal and external threads
- Lubrication grooves

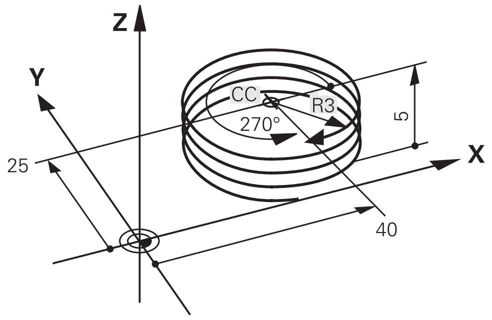
Dependencies of different thread shapes

The table shows the dependencies between machining direction, direction of rotation and radius compensation for the different thread shapes:

Internal thread	Work direction	Direction of rotation	Radius compensation
Right-handed	Z+	**DR+**	**RL**
	Z–	**DR-**	**RR**
Left-handed	Z+	**DR-**	**RR**
	Z–	**DR+**	**RL**

External thread	Work direction	Direction of rotation	Radius compensation
Right-handed	Z+	**DR+**	**RR**
	Z–	**DR-**	**RL**
Left-handed	Z+	**DR-**	**RL**
	Z–	**DR+**	**RR**

Programming a helix



> **i** Define the same algebraic sign for the direction of rotation **DR** and the incremental total angle **IPA**. The tool may otherwise move on a wrong path.

To program a helix:

C

- Select **C**

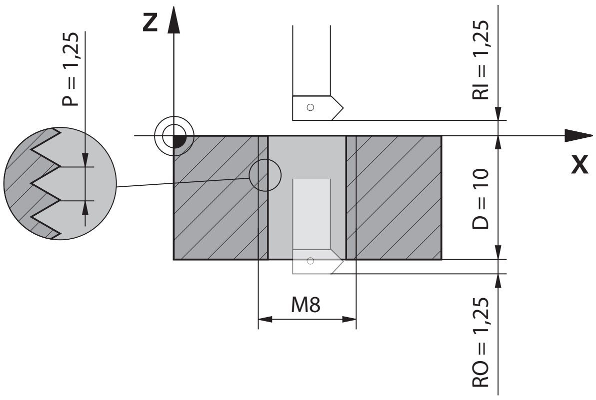
P

- Select **P**

I

- Select **I**
- Define the incremental total angle **IPA**
- Define the incremental total height **IZ**
- Select the direction of rotation
- Select radius compensation
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary

Example



This example includes the following default values:

- **M8** thread
- Left-handed thread miller

The drawing and the default values allow deriving the following information:

- Internal machining
- Right-hand thread
- **RR** radius compensation

The derived information requires the machining direction Z–.

Further information: "Dependencies of different thread shapes", Page 312

Specify and calculate the values below:

- Incremental total machining depth
- Number of thread grooves
- Incremental total angle

Formula	Definition
$IZ = D + RI + RO$	The incremental total machining depth **IZ** results from the thread depth **D** (depth) and from the optional thread run-in values **RI** (run-in) and thread run-out values **RO** (run-out).
$n = IZ \div P$	The number of thread grooves **n** (number) results from the incremental total machining depth **IZ** divided by the pitch **P** (pitch).
$IPA = n \times 360°$	The incremental total angle **IPA** results from the number of thread grooves **n** (number) multiplied by 360° for one complete revolution.

11 L Z+1,25 R0 FMAX	; Pre-position in the tool axis
12 L X+4 Y+0 RR F500	; Pre-position in the plane
13 CC X+0 Y+0	; Activate the pole
14 CP IPA-3600 IZ-12.5 DR-	; Cut the thread

Alternative solution with repetition of program parts

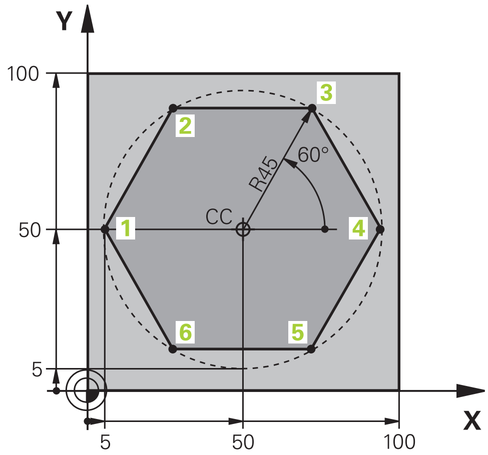
11 L Z+1.25	; Pre-position in the tool axis
12 L X+4 Y+0 RR F500	; Pre-position in the plane
13 CC X+0 Y+0	; Activate the pole
14 LBL 1	
15 CP IPA-360 IZ-1.25 DR-	; Cut the first thread groove
16 LBL CALL 1 REP 9	; Cut the next nine thread grooves, **REP 9** = number of remaining machining processes

The solution approach uses the thread pitch directly as the incremental plunging depth per revolution.

REP shows the number of necessary repeats required for reaching the calculated ten infeed runs.

Further information: "Subprograms and program section repeats with the label LBL", Page 332

11.4.7 Example: Polar straight lines



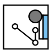
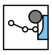
0 BEGIN PGM LINEARPO MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Workpiece blank definition
3 TOOL CALL 1 Z S4000	; Tool call
4 CC X+50 Y+50	; Define the datum for polar coordinates
5 L Z+250 R0 FMAX	; Retract the tool
6 LP PR+60 PA+180 R0 FMAX	; Pre-position the tool
7 L Z-5 R0 F1000 M3	; Move to working depth
8 APPR PLCT PR+45 PA+180 R5 RL F250	; Approach the contour at point 1 on a circular path with tangential connection
9 LP PA+120	; Move to point 2
10 LP PA+60	; Move to point 3
11 LP PA+0	; Move to point 4
12 LP PA-60	; Move to point 5
13 LP PA-120	; Move to point 6
14 LP PA+180	; Move to point 1
15 DEP PLCT PR+60 PA+180 R5 F1000	; Depart contour on a circular path with tangential connection
16 L Z+250 R0 FMAX M2	; Retract the tool, end program
17 END PGM LINEARPO MM	

11.5 Approaching and departing from a contour

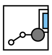
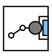
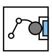
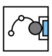
11.5.1 Overview of path shapes

Approach and departure functions are used to gently move the tool to or from a contour, without leaving any machining marks on the workpiece.

The **APPR** folder of the **Insert NC function** window contains the following functions:

Symbol	Function	Further information
	**APPR LT** or **APPR PLT** Use Cartesian or polar coordinates to approach a contour on a straight line with a tangential connection	Page 320
	**APPR LN** or **APPR PLN** Use Cartesian or polar coordinates to approach a contour on a straight line perpendicular to the first contour point	Page 321
	**APPR CT** or **APPR PCT** Use Cartesian or polar coordinates to approach a contour on a circular path with a tangential connection	Page 322
	**APPR LCT** or **APPR PLCT** Use Cartesian or polar coordinates to approach a contour on a circular path with a tangential connection and a straight line	Page 323

The **DEP** folder of the **Insert NC function** window contains the following functions:

Symbol	Function	Further information
	**DEP LT** Depart contour on a straight line with a tangential connection	Page 325
	**DEP LN** Depart contour on a straight line perpendicular to the last contour point	Page 325
	**DEP CT** Depart contour on a circular path with a tangential connection	Page 327
	**DEP LCT** or **DEP PLCT** Use Cartesian or polar coordinates to depart a contour on a circular path with a tangential connection and a straight line	Page 327

You can switch between entry of Cartesian and polar coordinates in the form or by pressing the **P** key.

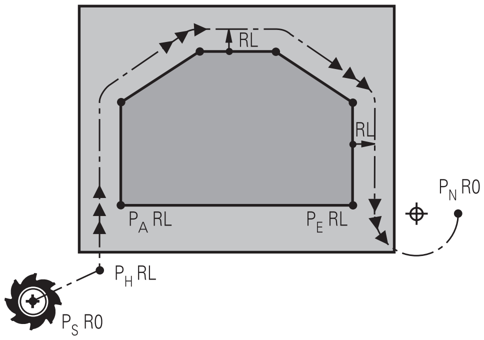
Further information: "Fundamentals of coordinate definitions", Page 290

Approaching or departing a helix

The tool approaches and departs a helix in the extension of the helix by moving on a circular path that connects tangentially to the contour. Use the **APPR CT** and **DEP CT** functions for this.

Further information: "Helix", Page 311

11.5.2 Positions for approach and departure



NOTICE

Danger of collision!

The control traverses from the current position (starting point P_S) to the auxiliary point P_H at the last feed rate entered. If you programmed **FMAX** in the last positioning block before the approach function, the control also approaches the auxiliary point P_H at rapid traverse.

- Program a feed rate other than **FMAX** before the approach function

The control uses the following positions when approaching and departing a contour:

- Starting point P_S

  You program the starting point P_S before an approach block without radius compensation. The starting point is located outside of the contour.
- Auxiliary point P_H

  For some path shapes an auxiliary point P_H is necessary for approaching or departing the contour. The control calculates the auxiliary point from the information in the approach or departure blocks.
- First contour point P_A

  You program the first contour point P_A in the approach block together with the radius compensation.
- Last contour point P_E

  You program the last contour point P_E with any path function.
- End point P_N

  The position P_N is located outside of the contour and results from the information in the departure block. The departure block automatically rescinds the radius compensation.

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning and incorrect auxiliary points P_H can also lead to contour damage. There is danger of collision during the approach movement!

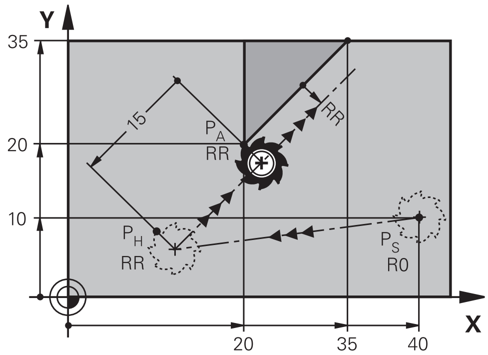
- Program a suitable pre-position
- Check the auxiliary point P_H, the sequence and the contour with the aid of the graphic simulation

11.5.3 Approach functions APPR LT and APPR PLT

Application

The control uses these functions to approach a contour on a straight line with tangential connection. You use **APPR LT** to define the starting point of the contour with Cartesian coordinates, and **APPR PLT** for polar coordinates.

Description of function



The control approaches the contour as follows:

- On a straight line from the starting point P_S to an auxiliary point P_H
- On a straight line, tangentially from the auxiliary point P_H to the first contour point P_A

  The auxiliary point P_H is separated from the first contour point P_A by the distance **LEN**.

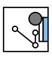
Programming APPR LT and APPR PLT

> If you want to program this path shape with polar coordinates, then you must first define a pole **CC**.
> **Further information:** "Polar coordinate datum at pole CC", Page 307

To define the approach function:

- Use any path function to approach the starting point P_S

- Select **APPR DEP**
- > The control opens the **Insert NC function** window.



- Select the path shape, for example **APPR LT**
- Define the coordinates of the first contour point P_A
- Use **LEN** to define the distance to the auxiliary point P_H
- Select radius compensation **RR/RL**

Example APPR LT

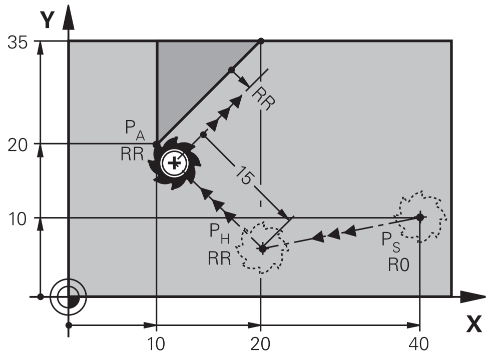
11 L X+40 Y+10 R0 FMAX M3	; Approach P_S without radius compensation
12 APPR LT X+20 Y+20 Z-10 LEN15 RR F100	; P_A with radius compensation RR; distance P_H to P_A: LEN 15
13 L X+35 Y+35	; End point of first contour element
14 L ...	; Next contour element

11.5.4 Approach functions APPR LN and APPR PLN

Application

The control uses these functions to approach a contour on a straight line perpendicular to the first contour point. You use **APPR LN** to define the starting point of the contour with Cartesian coordinates, and **APPR PLN** for polar coordinates.

Description of function



The control approaches the contour as follows:

- On a straight line from the starting point P_S to an auxiliary point P_H
- On a straight line, perpendicularly from the auxiliary point P_H to the first contour point P_A

  The auxiliary point P_H is separated from the first contour point P_A by the distance **LEN**.

> If you program **R0**, the control stops the machining/simulation with an error message.
> This behavior differs from the iTNC 530 control.

Programming APPR LN and APPR PLN

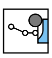
> If you want to program this path shape with polar coordinates, then you must first define a pole **CC**.
> **Further information:** "Polar coordinate datum at pole CC", Page 307

To define the approach function:

- Use any path function to approach the starting point P_S

APPR /DEP

- Select **APPR DEP**
- > The control opens the **Insert NC function** window.



- Select the path shape, for example **APPR LN**
- Define the coordinates of the first contour point P_A
- Use **LEN** to define the positive distance to the auxiliary point P_H
- Select radius compensation **RR/RL**

Example APPR LN

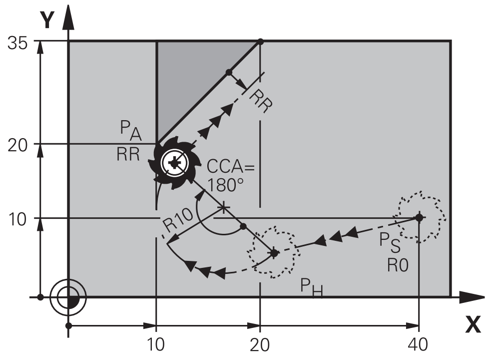
11 L X+40 Y+10 R0 FMAX M3	; Approach P_S without radius compensation
12 APPR LN X+10 Y+20 Z-10 LEN15 RR F100	; P_A with radius compensation RR; distance P_H to P_A: LEN 15
13 L X+20 Y+35	; End point of first contour element
14 L ...	; Next contour element

11.5.5 Approach functions APPR CT and APPR PCT

Application

The control uses these functions to approach a contour on a circular path with tangential connection. You use **APPR CT** to define the starting point of the contour with Cartesian coordinates, and **APPR PCT** for polar coordinates.

Description of function



The control approaches the contour as follows:

- On a straight line from the starting point P_S to an auxiliary point P_H
- On a circular path that tangentially transitions to the first contour element, from the auxiliary point P_H to the first contour point P_A

  The circular path from P_H to P_A is defined by center angle **CCA** and radius **R**. The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of radius **R**.

The table shows the relationship between the radius compensation, the algebraic sign of the radius **R**, and the direction of rotation:

Radius compensation	Algebraic sign of R	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Programming APPR CT and APPR PCT

> If you want to program this path shape with polar coordinates, then you must first define a pole **CC**.
>
> **Further information:** "Polar coordinate datum at pole CC", Page 307

To define the approach function:

- Use any path function to approach the starting point P_S

APPR /DEP

- Select **APPR DEP**
- > The control opens the **Insert NC function** window.

- Select the path shape, for example **APPR CT**
- Define the coordinates of the first contour point P_A
- Define the circle center angle **CCA**
- Define the radius **R** of the circular path
- Select radius compensation **RR/RL**

Notes

- If you enter a negative value for the radius **R**, the position of the auxiliary point P_H changes.
- If you program **R0**, the control stops the machining/simulation with an error message. This behavior differs from the iTNC 530 control.

Example APPR CT

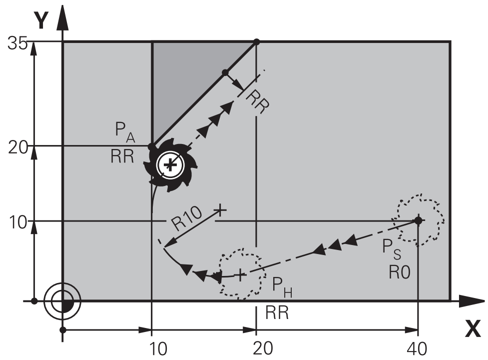
11 L X+40 Y+10 R0 FMAX M3	; Approach P_S without radius compensation
12 APPR CT X+10 Y+20 Z-10 CCA180 R+10 RR F100	; P_A with radius compensation RR, radius of the circular path: R 10
13 L X+20 Y+35	; End point of first contour element
14 L ...	; Next contour element

11.5.6 Approach functions APPR LCT and APPR PLCT

Application

The control uses these functions to approach a contour on a circular path with tangential connection and a straight line. You use **APPR LCT** to define the starting point of the contour with Cartesian coordinates, and **APPR PLCT** for polar coordinates.

Description of function



The control approaches the contour as follows:

- On a straight line from the starting point P_S to an auxiliary point P_H

  If you program the Z coordinate in the approach block, the control simultaneously moves the tool from the starting point P_S to the auxiliary point P_H.
- On a circular path in the working plane from the auxiliary point P_H to the first contour point P_A

  The circular path is connected tangentially both to the line P_S to P_H as well as to the first contour element. Once this information is entered, the radius **R** suffices to unambiguously define the tool path.

> The feed rate programmed in the approach block is effective for the entire path that the control traverses in the approach block. If no feed rate is programmed before the approach block, the control generates an error message.

Programming APPR LCT and APPR PLCT

> If you want to program this path shape with polar coordinates, then you must first define a pole **CC**.
>
> **Further information:** "Polar coordinate datum at pole CC", Page 307

To define the approach function:

- Use any path function to approach the starting point P_S

APPR /DEP

- Select **APPR DEP**
- The control opens the **Insert NC function** window.

- Select the path shape, for example **APPR LCT**
- Define the coordinates of the first contour point P_A
- Define a positive value for the radius **R** of the circular path
- Select radius compensation **RR/RL**

Example APPR LCT

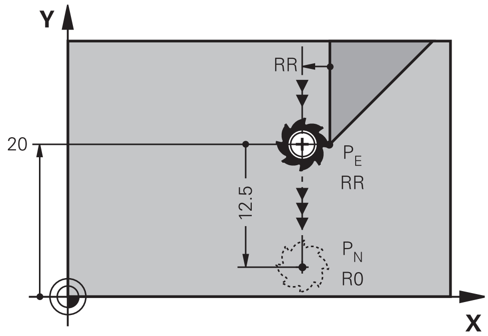
11 L X+40 Y+10 R0 FMAX M3	; Approach P_S without radius compensation
12 APPR LCT X+10 Y+20 Z-10 R10 RR F100	; P_A with radius compensation RR, radius of the circular path: R 10
13 L X+20 Y+35	; End point of first contour element
14 L ...	; Next contour element

11.5.7 Departure function DEP LT

Application

The control uses the **DEP LT** function to depart a contour on a straight line with tangential connection.

Description of function



The control departs the contour as follows:

- On a straight line from the last contour point P_E to the end point P_N
- The line lies on the extension of the last contour element.
  P_N is separated from P_E by the distance **LEN**.

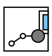
Programming DEP LT

To define the departure function:

- Program the last contour element with the end point P_E and radius compensation

APPR /DEP

- Select **APPR DEP**
- The control opens the **Insert NC function** window.



- Select **DEP LT**
- Use **LEN** to define the distance to the auxiliary point P_H

Example

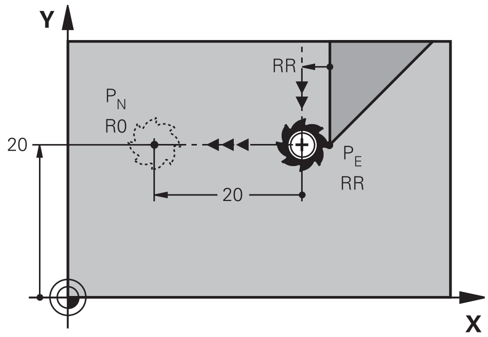
11 L Y+20 RR F100	; Last contour element P_E with radius compensation RR
12 DEP LT LEN12.5 F100	; Distance from P_E to P_N: LEN 12.5
13 L Z+100 FMAX M2	; Retract in Z, return jump, end program

11.5.8 Departure function DEP LN

Application

The control uses the **DEP LN** function to depart a contour on a straight line perpendicular to the last contour point.

Description of function



The control departs the contour as follows:

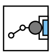
- On a straight line from the last contour point P_E to the end point P_N
- The line leads away from the last contour point P_E on a perpendicular path. P_N is separated from P_E by the distance **LEN** plus the tool radius.

Programming DEP LN

To define the departure function:

- Program the last contour element with the end point P_E and radius compensation

- Select **APPR DEP**
- The control opens the **Insert NC function** window.



- Select **DEP LN**
- Use **LEN** to define the positive distance to the auxiliary point P_H

Example

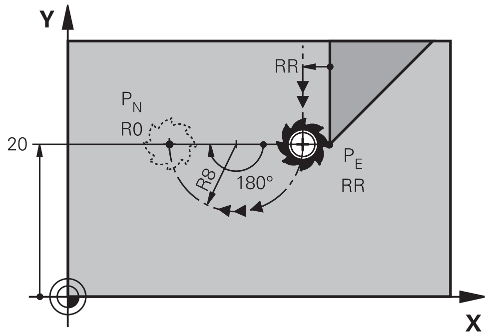
11 L Y+20 RR F100	; Last contour element P_E with radius compensation RR
12 DEP LN LEN+20 F100	; Distance from P_E to P_N: LEN 20
13 L Z+100 FMAX M2	; Retract in Z, return jump, end program

11.5.9 Departure function DEP CT

Application

The control uses the **DEP CT** function to depart a contour on a circular path with tangential connection.

Description of function



The control departs the contour as follows:

- On a circular path from the last contour point P_E to the end point P_N
- The circular path connects tangentially to the last contour element
  The circular path from P_E to P_N is defined by center angle **CCA** and radius **R**. The direction of rotation of the circular path depends on the active radius compensation and the algebraic sign of radius **R**.

The table shows the relationship between the radius compensation, the algebraic sign of the radius **R**, and the direction of rotation:

Radius compensation	Algebraic sign of R	Direction of rotation
RL	Positive	Counterclockwise
RL	Negative	Clockwise
RR	Positive	Clockwise
RR	Negative	Counterclockwise

The following applies regarding the center angle **CCA**:

- Only positive input values
- Maximum input value 360°

Note

If you enter a negative value for the radius **R**, the position of the end point P_N changes.

Example

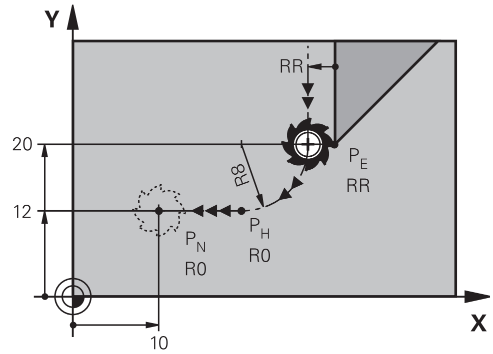
11 L Y+20 RR F100	; Last contour element P_E with radius compensation RR
12 DEP CT CCA 180 R+8 F100	; Center angle CCA 180°, radius of the circular path: R 8
13 L Z+100 FMAX M2	; Retract in Z, return jump, end program

11.5.10 Departure functions DEP LCT and DEP PLCT

Application

The control uses these functions to depart a contour on a circular path with tangential connection and a straight line. You use **DEP LCT** to define the end point with Cartesian coordinates, and **DEP PLCT** for polar coordinates.

Description of function



The control departs the contour as follows:

- On a circular path from the last contour point P_E to the auxiliary point P_H
- On a straight line from the auxiliary point P_H to the end point P_N

  If you program the Z coordinate in the departure block, the control simultaneously moves the tool from the auxiliary point P_H to the end point P_N.

  The circular path is connected tangentially both to the last contour element as well as to the line P_H to P_N. Once this information is entered, the radius **R** suffices to unambiguously define the tool path.

Programming DEP LCT and DEP PLCT

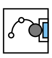
> If you want to program this path shape with polar coordinates, then you must first define a pole **CC**.
>
> **Further information:** "Polar coordinate datum at pole CC", Page 307

To define the departure function:

- ▶ Program the last contour element with the end point P_E and radius compensation

APPR /DEP

- ▶ Select **APPR DEP**
- > The control opens the **Insert NC function** window.



- ▶ Select the path shape, for example **DEP LN**
- ▶ Define the coordinates of the end point P_N
- ▶ Define a positive value for the radius **R** of the circular path

Example DEP LCT

11 L Y+20 RR F100	; Last contour element P_E with radius compensation RR
12 DEP LCT X+10 Y+12 R+8 F100	; Coordinates P_N, radius of the circular path: R 8
13 L Z+100 FMAX M2	; Retract in Z, return jump, end program

Definitions

Abbreviation	Definition
APPR (approach)	Approach function
DEP (departure)	Departure function
L (line)	Line segment
C (circle)	Circle
T (tangential)	Continuous, smooth transition
N (normal)	Perpendicular line

12

Programming Techniques

12.1 Subprograms and program section repeats with the label LBL

Application

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as necessary. Use subprograms to insert contours or complete machining steps after the end of the program and call them in the NC program. Program section repeats repeat single or several NC blocks during the NC program. Subprograms and program section repeats can also be combined.

Subprograms and program section repeats are programmed with the NC function **LBL**.

Related topics

- Executing NC programs within another NC program
  Further information: "Calling an NC program with PGM CALL", Page 336
- Jumps with conditions as if-then decisions.
  Further information: " Jump commands folder", Page 1283

Description of function

The label **LBL** is used for defining the machining steps for subprograms and program section repeats.

The control offers the following keys and icons in connection with labels:

Key or icon	Function
LBL SET	Create **LBL**
LBL CALL	Call **LBL**: Jump to the label in the NC program
	In case of **LBL** number: Enter the next free number automatically

Defining a label with LBL SET

The **LBL SET** function defines a new label in the NC program.

Each label must be unambiguously identifiable in the NC program by a number or a name. If a number or a name exists twice in an NC program, the control shows a warning ahead of the NC block.

LBL 0 marks the end of a subprogram. This number is the only one which may exist any number of times in the NC program.

Input

11 LBL "Reset"	; Subprogram for resetting a coordinate transformation
12 TRANS DATUM RESET	
13 LBL 0	

The NC function includes the following syntax elements:

Syntax element	Meaning
LBL	Syntax initiator for a label
0 or **" "**	Number or name of the label. Fixed or variable number or name Input: **0...65535** or **text width 32** Use an icon to enter the next free number automatically. **Further information:** "Description of function", Page 332

Calling a label with CALL LBL

The **CALL LBL** function calls a label in the NC program.

When the control reads **CALL LBL**, it jumps to the defined label and continues executing the NC program from this NC block. When the control reads **LBL 0**, it jumps back to the next NC block after **CALL LBL**.

In case of program section repeats, you can optionally define that the control executes that jump several times.

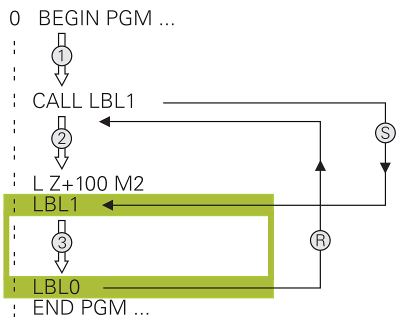
Input

11 CALL LBL 1 REP2	; Call label 1 twice

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL LBL	Syntax initiator for calling a label
Number, **" "** or **QS**	Number or name of the label. Fixed or variable number or name Input: **1...65535** or **text width 32** or **0...1999** The label can be selected from a selection menu including all labels available in the NC program.
REP	Number of repetitions until the control executes the next NC block Optional syntax element

Subprograms



A subprogram allows calling parts of an NC program any number of times at different points of the NC program, e. g. machining positions or a contour.

A subprogram starts with a **LBL**label and ends with **LBL 0**. **CALL LBL** calls the subprogram from any point in the NC program. In this process, repetitions must not be defined with **REP**.

The control executes the NC program as follows:

1. The control executes the NC program up to the **CALL LBL** function.
2. The control jumps to the beginning of the defined subprogram **LBL**.
3. The control executes the subprogram up to the subprogram end **LBL 0**.
4. After that, the control jumps to the next NC block after **CALL LBL** and continues executing the NC program.

The following conditions apply to subprograms:

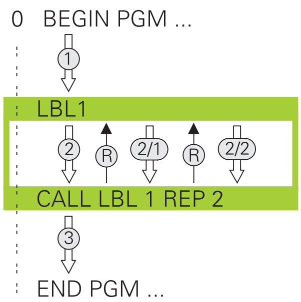
- A subprogram cannot call itself
- **CALL LBL 0** is not permitted (Label 0 is only used to mark the end of a subprogram).
- Write subprograms after the NC block with M2 or M30

  If subprograms are located in the NC program before the NC block with M2 or M30, they will be executed at least once even if they are not called

The control displays information about the active subprogram on the **LBL** tab of the **Status** workspace.

Further information: "LBL tab", Page 153

Program-section repeats



A program section repeat allows repeating a part of an NC program any number of times, e. g. contour machining with incremental infeed.

A program section repeat starts with a **LBL** label and ends after the last programmed repetition **REP** of the label call **CALL LBL**.

The control executes the NC program as follows:

1 The control executes the NC program up to the **CALL LBL** function.

  In this process, the control already executes the program section once because the program section to be repeated is positioned ahead of the **CALL LBL** function.

2 The control jumps to the beginning of the program section repeat **LBL**.

3 The control repeats the program section as many times as programmed under **REP**.

4 After that, the control continues executing the NC program.

The following conditions apply to program section repeats:

- Program the program section repeat before the end of the program with **M30** or **M2**.
- No **LBL 0** can be defined with a program section repeat.
- The total number of times the program section is executed is always one more than the programmed number of repeats, because the first repeat starts after the first machining process.

The control displays information about the active program section repeat on the **LBL** tab of the **Status** workspace.

Further information: "LBL tab", Page 153

Notes

- The control displays the NC function **LBL SET** in the structure by default.

  Further information: "Structure column in the Program workspace", Page 1386
- You can repeat a program section up to 65 534 times in succession
- The following characters are allowed in the name of a label: # $ % & , - _ . 0 1 2 3 4 5 6 7 8 9 @ a b c d e f g h i j k l m n o p q r s t u v w x y z- A B C D E F G H I J K L M N O P Q R S T U V W X Y Z
- The following characters are not allowed in the name of a label: <blank> ! " ‘ ( ) * + : ; < = > ? [/] ^ ` { | } ~
- Before creating your NC program, compare the subprogram and program section repeat programming techniques using if-then decisions.

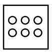
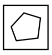
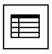
  You can thereby avoid possible misunderstandings and programming errors.

  Further information: " Jump commands folder", Page 1283

12.2 Selection functions

12.2.1 Overview of selection functions

The **Selection functions** folder of the **Insert NC function** window contains the following functions:

Icon	Function	Further information
	Call an NC program with **PGM CALL**	Page 336
	Select a datum table with **SEL TABLE**	Page 958
	Select a point table with **SEL PATTERN**	Page 347
	Select a contour program with **SEL CONTOUR**	Page 359
	Select an NC program with **SEL PGM**	Page 338
	Call the last selected file with **CALL SELECTED PGM**	Page 338
	Select any NC program with **SEL CYCLE** as a machining cycle	Page 425
	Select a correction table with **SEL CORR-TABLE**	Page 1044
	Open the file with **OPEN FILE**	Page 1079

12.2.2 Calling an NC program with PGM CALL

Application

The **PGM CALL** function calls another separate NC program from an NC program. The control executes the called NC program at the point where you called it in the NC program. This allows executing e. g. a machining process with different transformations.

Related topics

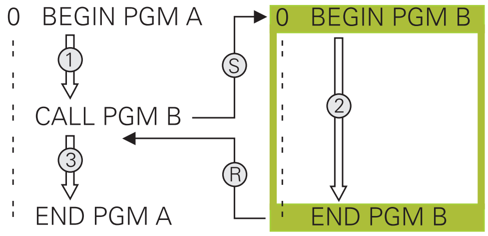
- Program call with cycle **12 PGM CALL**

  Further information: "Cycle 12 PGM CALL ", Page 341
- Program call following selection

  Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 338
- Executing several NC programs as a job list

  Further information: "Pallet Machining and Job Lists", Page 1763

Description of function



The control executes the NC program as follows:

1 The control executes the calling NC program until another NC program is called with **CALL PGM**.
2 After that, the control executes the called NC program up to the last NC block.
3 Then the control continues executing the calling NC program from the next NC block after **CALL PGM**.

The following conditions apply to program calls:

- The called NC program must not contain a **CALL PGM** call into the calling NC program. This creates an endless loop.
- The called NC program must not contain a miscellaneous function **M30** or **M2**. If you have defined subprograms with labels in the called NC program, you can replace **M30** or **M2** by the jump function **FN 9: If +0 EQU +0 GOTO LBL 99**. Now the control will not execute e. g. subprograms without a call.

  Further information: "Unconditional jump", Page 1284

  If the called NC program contains the miscellaneous functions, the control generates an error message.
- The called NC program must be complete. If the NC block **END PGM** is missing, the control outputs an error message.

Input

11 CALL PGM reset.h	Call NC program

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL PGM	Syntax initiator for calling an NC program
reset.h	Path of the called NC program The NC program can be selected in a selection menu.

Notes

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. If you do not specifically rescind the coordinate transformations in the called NC program, these transformations will also take effect in the calling NC program. Danger of collision during machining!

- Reset used coordinate transformations in the same NC program
- Check the machining sequence using a graphic simulation if required

- The program call path including the name of the NC program may contain no more than 255 characters.
- If the called file is located in the same directory as the file you are calling it from, you can also enter just the file name without the path. If you select the file using the selection menu, the control automatically proceeds in this manner.
- If you want to program variable program calls in connection with string parameters, use the **SEL PGM** function.

  Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 338
- As a rule, Q parameters are effective globally with a **PGM CALL**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program. If required, use QL parameters which are effective only in the active NC program.
- While the control is executing the calling NC program, editing of all called NC programs is disabled.

12.2.3 Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM

Application

The function **SEL PGM** allows selecting another separate NC program that you can call at a different position in the active NC program. The control executes the selected NC program at the position where you call it in the calling NC program using **CALL SELECTED PGM**.

Related topics

- Calling the NC program directly

  Further information: "Calling an NC program with PGM CALL", Page 336

Description of function

The control executes the NC program as follows:

1 The control executes the NC program until another NC program is called with **CALL PGM**. When the control reads **SEL PGM**, it remembers the defined NC program.
2 When the control reads **CALL SELECTED PGM**, it calls the NC program previously selected at this point.
3 After that, the control executes the called NC program up to the last NC block.
4 Then the control continues executing the calling NC program with the next NC block after **CALL SELECTED PGM**.

The following conditions apply to program calls:

- The called NC program must not contain a **CALL PGM** call into the calling NC program. This creates an endless loop.
- The called NC program must not contain a miscellaneous function **M30** or **M2**. If you have defined subprograms with labels in the called NC program, you can replace **M30** or **M2** by the jump function **FN 9: If +0 EQU +0 GOTO LBL 99**. Now the control will not execute e. g. subprograms without a call.

  Further information: "Unconditional jump", Page 1284

  If the called NC program contains the miscellaneous functions, the control generates an error message.
- The called NC program must be complete. If the NC block **END PGM** is missing, the control outputs an error message.

Input

11 SEL PGM "reset.h"	; Select an NC program for calling
*** - ...**	
21 CALL SELECTED PGM	; Call the selected NC program

The NC function **SEL PGM** includes the following syntax elements:

Syntax element	Meaning
SEL PGM	Syntax initiator for selecting an NC program to be called
" " or **QS**	Path of the called NC program Fixed or variable name The NC program can be selected in a selection menu.

The NC function **CALL SELECTED PGM** includes the following syntax elements:

Syntax element	Meaning
CALL SELECTED PGM	Syntax for calling the selected NC program

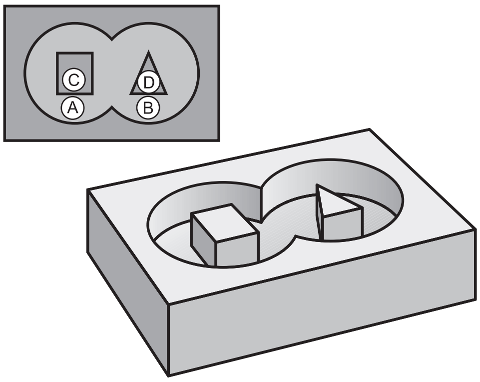
Notes

- Within the **SEL PGM** function, the NC program can also be selected with QS parameters so that the program call can be variably controlled.
- If an NC program called by **CALL SELECTED PGM** is missing, the control interrupts the execution or simulation of the program with an error message. In order to avoid undesired interruptions during program run, you can use the function **FN 18: SYSREAD** (**ID10 NR110** and **NR111**) to check all paths at the beginning of the program.

  Further information: "Read system data with FN 18: SYSREAD", Page 1290
- If the called file is located in the same directory as the file you are calling it from, you can also enter just the file name without the path. If you select the file using the selection menu, the control automatically proceeds in this manner.
- As a rule, Q parameters are effective globally with a **PGM CALL**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program. If required, use QL parameters which are effective only in the active NC program.
- While the control is executing the calling NC program, editing of all called NC programs is disabled.

12.3 Cycle 14 CONTOUR

Application



In Cycle **14 CONTOUR**, list all subprograms that are to be superimposed to define the overall contour .

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Cycle **14** is DEF-active which means that it becomes effective as soon as it is defined in the NC program.
- You can list up to 12 subprograms (subcontours) in Cycle **14**.

12.3.1 Cycle parameters

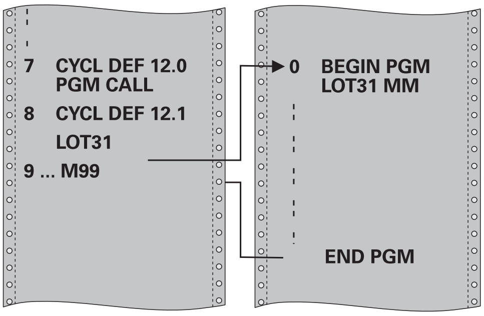
Help graphic	Parameter
	Label numbers for contour? Enter all label numbers for the individual subprograms that are to be superimposed to define a contour. Confirm each number with the ENT key. Confirm your entries with the **END** key. Up to 12 subprogram numbers are possible. Input: **0...65535**

Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL1 /2
```

12.4 Cycle 12 PGM CALL

Application



NC programs that you have created (such as special drilling cycles or geometrical modules) can be written as machining cycles. These NC programs can then be called like normal cycles.

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- As a rule, Q parameters are globally effective when called with Cycle **12**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program.

Notes on programming

- The NC program you are calling must be stored in the internal memory of your control.
- If the NC program you are defining to be a cycle is located in the same directory as the NC program you are calling it from, you need only enter the program name.
- If the NC program you are defining to be a cycle is not located in the same directory as the NC program you are calling it from, you must enter the complete path, for example **TNC:\KLAR35\FK1\50.H**.

12.4.1 Cycle parameters

Help graphic	Parameter
	Program name Enter the name of the NC program to be called and, if necessary, the path where it is located, Use the Select File Select in the action bar of the NC program to be called.

Call the NC program with:

- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Declare NC program 1_Plate.h as a cycle and call it with M99

```
11 CYCL DEF 12.0 PGM CALL
12 CYCL DEF 12.1 PGM TNC:\nc_prog\demo\OCM\1_Plate.h
13 L  X+20  Y+50 R0 FMAX M99
```

12.5 Nesting of programming techniques

Application

Programming techniques can also be combined with one another, e. g. another separate NC program or a subprogram can be called in a program section repeat.

The nesting depth defines, among other things, how often program sections or subprograms may contain further subprograms or program section repeats.

Related topics

- Subprograms
  Further information: "Subprograms", Page 334
- Program section repeats
  Further information: "Program-section repeats", Page 335
- Calling a separate NC program
  Further information: "Selection functions", Page 336

Description of function

The following maximum nesting depths apply to NC programs:

- Maximum nesting depth for subprograms: 19
- Maximum nesting depth for external NC programs: 19, for which a **CYCL CALL** has the effect of calling an external program
- You can nest program section repeats as often as desired

12.5.1 Example

Subprogram call within a subprogram

0 BEGIN PGM UPGMS MM	
*** - ...**	
11 CALL LBL “UP1“	; Call subprogram **LBL "UP1"**
*** - ...**	
21 L Z+100 R0 FMAX M30	; Last program block of main program with M30
22 LBL “UP1“	; Start of subprogram **"UP1"**
*** - ...**	
31 CALL LBL 2	; Call subprogram **LBL 2**
*** - ...**	
41 LBL 0	; End of sub program **"UP1"**
42 LBL 2	; Start of subprogram **LBL 2**
*** - ...**	
51 LBL 0	; End of subprogram **LBL 2**
52 END PGM UPGMS MM	

The control executes the NC program as follows:

1. NC program UPGMS is executed up to NC block 11.
2. Subprogram UP1 is called and executed up to NC block 31.
3. Subprogram 2 is called, and executed up to NC block 51. End of subprogram 2 and return jump to the subprogram from which it was called.
4. Subprogram UP1 is called, and executed from NC block 32 up to NC block 41. End of subprogram UP1 and return jump to NC program UPGMS.
5. NC program UPGMS is executed from NC block 12 up to NC block 21. Program end with return jump NC block 1.

Program-section repeat within a program section repeat

0 BEGIN PGM REPS MM	
* - ...	
11 LBL 1	; Start of program section 1
* - ...	
21 LBL 2	; Start of program section 2
* - ...	
31 CALL LBL 2 REP 2	; Call program section 2 and repeat twice
* - ...	
41 CALL LBL 1 REP 1	; Call program section 1 including program section 2 and repeat once
* - ...	
51 END PGM REPS MM	

The control executes the NC program as follows:

1 NC program REPS is executed up to NC block 31.
2 The program section between NC block 31 and NC block 21 is repeated twice, meaning that it is executed three times in total.
3 NC program REPS is executed from NC block 32 up to NC block 41.
4 The program section between NC block 41 and NC block 11 is repeated once, meaning that it is executed twice in total (including the program section repeat between NC block 21 and NC block 31).
5 NC program REPS is executed from NC block 42 up to NC block 51. Program end with return jump to NC block 1.

Subprogram call within a program section repeat

0 BEGIN PGM UPGREP MM	
* - ...	
11 LBL 1	; Start of program section 1
12 CALL LBL 2	; Call subprogram 2
13 CALL LBL 1 REP 2	; Call program section 1 and repeat twice
* - ...	
21 L Z+100 R0 FMAX M30	; Last NC block of main program with M30
22 LBL 2	; Start of subprogram 2
* - ...	
31 LBL 0	; End of subprogram 2
32 END PGM UPGREP MM	

The control executes the NC program as follows:

1 NC program UPGREP is executed up to NC block 12.
2 Subprogram 2 is called, and executed up to NC block 31.
3 The program section between NC block 13 and NC block 11 (including subprogram 2) is repeated twice, meaning that it is executed three times in total.
4 NC program UPGREP is executed from NC block 14 up to NC block 21. Program end with return jump to NC block 1.

13

Contour and Point Definitions

13.1 Point tables

Application

With a point table you can execute one or more cycles in sequence on an irregular point pattern.

Related topics

- Contents of a point table, hiding individual points
  Further information: "Point table", Page 1856

Description of function

Coordinates in a point table

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting point coordinates of the respective cycle, e.g. center coordinates of a circular pocket. The coordinates of the spindle axis correspond to the coordinate of the workpiece surface.

The control retracts the tool to the clearance height when traversing between the starting points. Depending on which is greater the control uses either the tool axis coordinate from the cycle call or the value from cycle parameter **Q204 2ND SET-UP CLEARANCE**.

NOTICE

Danger of collision!

If you program a clearance height for individual points in a point table, the control will ignore the value from the cycle parameter **Q204 2ND SET-UP CLEARANCE** for all points!

- Program the function **GLOBAL DEF 125 POSITIONING** so that the control will take into account the clearance height only for the respective point.

Effect with cycles

SL cycles and Cycle 12

The control interprets the points in the point table as an additional datum shift.

Cycles 200 to 208, 262 to 267

The control interprets the points of the working plane as coordinates of the hole centers. If you want to use the coordinate defined in the point table as the starting point coordinate in the tool axis, you must define the coordinate of the workpiece upper edge (**Q203**) as 0.

Cycles 210 to 215

The control interprets the points as an additional datum shift. If you want to use the points defined in the point table as the starting point coordinates, you must program the starting points and the coordinate of the workpiece upper edge (**Q203**) in the respective milling cycle as 0.

You can no longer insert these cycles on the control, but you can edit and run them in existing NC programs.

Cycles 251 to 254

The control interprets the points on the working plane as coordinates of the cycle starting point. If you want to use the coordinate defined in the point table as the starting point coordinate in the tool axis, you must define the coordinate of the workpiece upper edge (**Q203**) as 0.

13.1.1 Selecting the point table in the NC program with SEL PATTERN

To select the point table:

Insert NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **SEL PATTERN**
- ▶ Select **File selection**
- > The control opens a window for the file selection.
- ▶ Select the desired point table through the file structure
- ▶ Confirm your input
- > The control concludes the NC block.

If the point table is not stored in the same directory as the NC program, you must define the complete path name. In the **Program settings** window you can define whether the control creates absolute or relative paths.

Further information: "Settings in the Program workspace", Page 197

Example

```
7 SEL PATTERN  "TNC:\nc_prog\Positions.PNT
```

13.1.2 Calling the cycle with a point table

If you want to call a cycle at the points that you defined in the point table, then program the cycle call with **CYCLE CALL PAT**.

CYCL CALL PAT enables the control to execute the point table that you defined last.

To call a cycle in conjunction with a point table:

Insert NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.

CYCL CALL

- ▶ Select **CYCL CALL PAT**
- ▶ Enter a feed rate

> The control will use this feed rate to traverse between the points of the point table. If you do not enter a feed rate, the control moves the tool at the feed rate last defined.

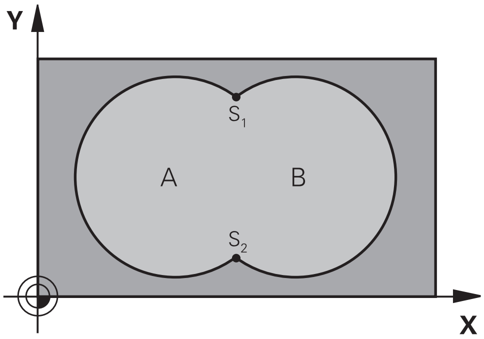
- ▶ Define miscellaneous functions, if necessary
- ▶ Confirm your input with the **END** key

Notes

- In the **GLOBAL DEF 125** function you can use the setting **Q435**=**1** to force the control to always move to the 2nd set-up clearance from the cycle during the positioning between the points.
- If you want to move at reduced feed rate when pre-positioning in the tool axis, program the **M103** miscellaneous function.
- With **CYCL CALL PAT** the control runs the point table that you last defined, even if you defined the point table with an NC program that was nested with **CALL PGM**.

13.2 Superimposed contours

13.2.1 Fundamentals



Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

13.2.2 Subprograms: overlapping pockets

> The following examples show contour subprograms that are called by Cycle **14 CONTOUR** in a main program.

Pockets A and B overlap.

The control calculates the points of intersection S1 and S2. They need not be programmed.

The pockets are programmed as full circles.

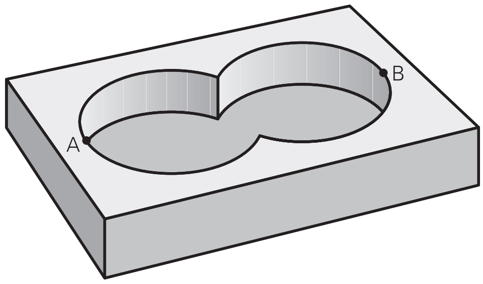
Subprogram 1: Pocket A

```
11 LBL 1
12 L  X+10  Y+10 RR
13 CC  X+35  Y+50
14 C  X+10  Y+50 DR-
15 LBL 0
```

Subprogram 2: Pocket B

16 LBL 2
17 L X+90 Y+50 RR
18 CC X+65 Y+50
19 C X+90 Y+50 DR-
20 LBL 0

13.2.3 Surface resulting from sum



Both surfaces A and B are to be machined, including the overlapping area:

- The surfaces A and B must be pockets
- The first pocket (in Cycle **14**) must start outside the second pocket

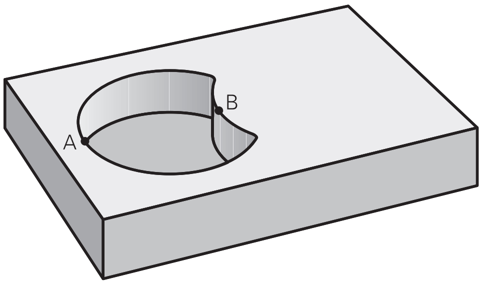
Surface A:

11 LBL 1
12 L X+10 Y+50 RR
13 CC X+35 Y+50
14 C X+10 Y+50 DR-
15 LBL 0

Surface B:

16 LBL 2
17 L X+90 Y+50 RR
18 CC X+65 Y+50
19 C X+90 Y+50 DR-
20 LBL 0

13.2.4 Surface resulting from difference



Surface A is to be machined without the portion overlapped by B:

- Surface A must be a pocket and B an island.
- A must start outside of B.
- B must start inside of A.

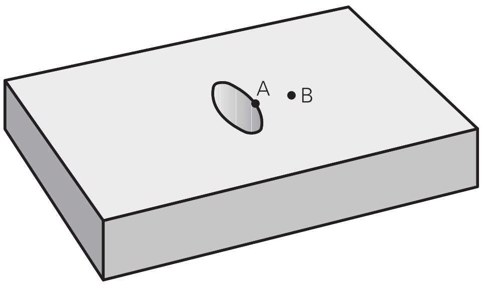
Surface A:

11 LBL 1
12 L X+10 Y+50 RR
13 CC X+35 Y+50
14 C X+10 Y+50 DR-
15 LBL 0

Surface B:

16 LBL 2
17 L X+40 Y+50 RL
18 CC X+65 Y+50
19 C X+40 Y+50 DR-
20 LBL 0

13.2.5 Surface resulting from intersection



Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- A and B must be pockets
- A must start inside of B

Surface A:

```
11 LBL 1
12 L  X+60  Y+50 RR
13 CC  X+35  Y+50
14 C  X+60  Y+50 DR-
15 LBL 0
```

Surface B:

```
16 LBL 2
17 L  X+90  Y+50 RR
18 CC  X+65  Y+50
19 C  X+90  Y+50 DR-
20 LBL 0
```

13.3 Simple contour formula

13.3.1 Fundamentals

Program structure: Machining with SL Cycles and simple contour formula

```
0 BEGIN CONTDEF MM
...
5 CONTOUR DEF
...
6 CYCL DEF 20 CONTOUR DATA
...
8 CYCL DEF 21 ROUGH-OUT
...
9 CYCL CALL
...
13 CYCL DEF 23 FLOOR FINISHING
...
14 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING
...
17 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 END PGM CONTDEF MM
```

Using simple contour formulas, you can easily combine up to nine subcontours (pockets or islands) to program a particular contour. The control calculates the complete contour from the selected subcontours.

> The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** contour elements.

Void areas

Using optional void areas **V** (**void**), you can exclude areas from machining. These area can be, for example, contours in castings or areas machined in previous steps. You can define up to five void areas.

If you are using OCM cycles, the control will plunge vertically within void areas.

If you are using SL Cycles **22** to **24**, the control will determine the plunging position, regardless of any defined void areas.

Run the simulation to verify proper behavior.

Properties of the subcontours

- Do not program radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are permitted; if they are programmed within the subcontours, they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored..
- The working plane is defined in the first coordinate block of the subprogram.

Cycle properties

- The control automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions; the cutter traverses around islands instead of over them.
- The radius of inside corners can be programmed; the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations).
- The contour is approached on a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

The machining dimensions, such as milling depth, allowances, and clearance height, can be entered centrally in Cycle **20 CONTOUR DATA** or **271 OCM CONTOUR DATA**.

13.3.2 Entering a simple contour formula

You can use the selection possibility in the action bar or in the form to interlink various contours in a mathematical formula.

Proceed as follows:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **CONTOUR DEF**
- > The control opens the dialog for entering the contour formula.
- Enter the first subcontour **P1**
- Select the **P2** pocket or **I2** island selection possibility
- Enter second subcontour
- If needed, enter the depth of the second subcontour.
- > Carry on with the dialog as described above until you have entered all subcontours.
- Define void areas **V** as needed

> The depth of the void areas corresponds to the total depth that you define in the machining cycle.

You can enter contours in the following ways:

Possible setting		Function
File	■ Input ■ File selection	Define the name of the contour or select File Selection
QS		Define the number of a QS parameter
LBL	■ Number ■ Name ■ QS	Define the number, name or QS parameter for a label

Example:

```
11 CONTOUR DEF P1 = LBL 1 I2 = LBL 2 DEPTH5 V1 = LBL 3
```

> Programming notes:
> - The first depth of the subcontour is the cycle depth. This is the maximum depth for the programmed contour. Other subcontours cannot be deeper than the cycle depth Therefore, always start programming the subcontour with the deepest pocket.
> - If the contour is defined as an island, the control interprets the entered depth as the island height. The entered value (without an algebraic sign) then refers to the workpiece top surface!
> - If you enter a value of 0 for the depth, then the depth defined in Cycle **20** is effective for pockets. For islands, this means that they extend up to the workpiece surface!
> - If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path.

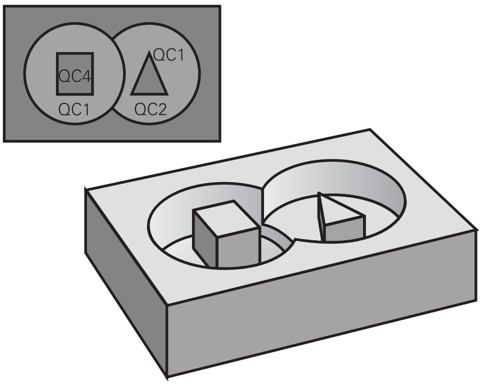
13.3.3 Machining contours with SL or OCM cycles

> The defined entire contour is machined with the SL cycles or the OCM cycles Page 452.

13.4 Complex contour formula

13.4.1 Fundamentals

Using complex contour formulas, you can combine several subcontours (pockets or islands) to program complex contours. You define the individual subcontours (geometry data) in separate NC programs. In this way, any subcontour can be reused any number of times. The control calculates the complete contour from the selected subcontours, which you link through a contour formula.



Program structure: Machining with SL Cycles and complex contour formula

```
0 BEGIN CONT MM
...
5 SEL CONTOUR "MODEL"
6 CYCL DEF 20 CONTOUR DATA
...
8 CYCL DEF 21 ROUGH-OUT
...
9 CYCL CALL
...
13 CYCL DEF 23 FLOOR FINISHING
...
14 CYCL CALL
...
16 CYCL DEF 24 SIDE FINISHING
...
17 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 END PGM CONT MM
```

Programming notes:

- The memory capacity for programming an SL cycle (all contour description programs) is limited to **128 contours.** The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. You can program up to **16384** contour elements.
- To use SL cycles with contour formulas, it is mandatory that your program is structured carefully. These cycles enable you to save frequently used contours in individual NC programs. Using the contour formula, you can connect the subcontours to define a complete contour and specify whether it applies to a pocket or island.

Properties of the subcontours

- The control assumes that each contour is a pocket. Thus, do not program a radius compensation.
- The control ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are permitted—if they are programmed within the subcontours, they are also effective in the NC programs called subsequently. However, they need not be reset after the cycle call.
- Although the called NC programs can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the NC program.
- Subcontours can be defined with different depths according to your requirements.

Cycle properties

- The control automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions; the cutter traverses around islands instead of over them.
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane)
- The contour is machined throughout in either climb or up-cut milling

The machining dimensions, such as milling depth, allowances, and clearance height, can be entered centrally in Cycle **20 CONTOUR DATA** or **271 OCM CONTOUR DATA**.

Program structure: Calculation of the subcontours with contour formula

```
0 BEGIN MODEL MM
1 DECLARE CONTOUR QC1 = "120"
2 DECLARE CONTOUR QC2 = "121" DEPTH15
3 DECLARE CONTOUR QC3 = "122" DEPTH10
4 DECLARE CONTOUR QC4 = "123" DEPTH5
5 QC10 = ( QC1 | QC3 | QC4 ) \ QC2
6 END PGM MODEL MM
```

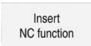
```
0 BEGIN PGM 120 MM
1 CC X+75 Y+50
2 LP PR+45 PA+0
3 CP IPA+360 DR+
4 END PGM 120 MM
```

```
0 BEGIN PGM 121 MM
...
```

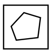
13.4.2 Selecting an NC program with contour definition

With the **SEL CONTOUR** function, you select an NC program with contour definitions, from which the control extracts the contour descriptions:

Proceed as follows:



- Select **Insert NC function**
- > The control opens the **Insert NC function** window.



- Select **SEL CONTOUR**
- > The control opens the dialog for entering the contour formula.
- Definition of the contour

You can enter contours in the following ways:

Possible setting		Function
File	■ Input ■ File selection	Define the name of the contour or select File Selection
QS		Define the number of a string parameter

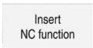
Programming notes:

- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path.
- Program a **SEL CONTOUR** block before the SL cycles. Cycle **14 CONTOUR** is no longer necessary if you use **SEL CONTOUR**.

13.4.3 Defining a contour description

Using the **DECLARE CONTOUR** function in your NC program, you enter the path for NC programs from which the control extracts the contour descriptions. In addition, you can select a separate depth for this contour description.

Proceed as follows:



- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **DECLARE CONTOUR**
- > The control opens the dialog for entering the contour formula.
- Enter the number for the contour designator **QC**
- Defining a contour description

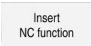
You can enter contours in the following ways:

Possible setting		Function
File	■ Input ■ File selection	Define the name of the contour or select File Selection
QS		Define the number of a string parameter

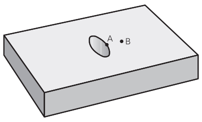
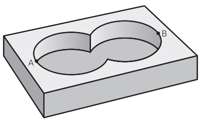
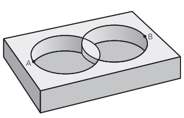
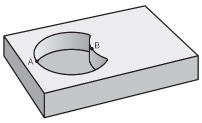
Programming notes:
- With the entered contour designators **QC** you can include the various contours in the contour formula.
- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path.
- If you program separate depths for contours, then you must assign a depth to all subcontours (assign the depth 0 if necessary).
- The control will only take different depths (**DEPTH**) into account if the elements overlap. In case of pure islands inside a pocket, this is not the case. Use a simple contour formula for this purpose.
  Page 352

13.4.4 Entering a complex contour formula

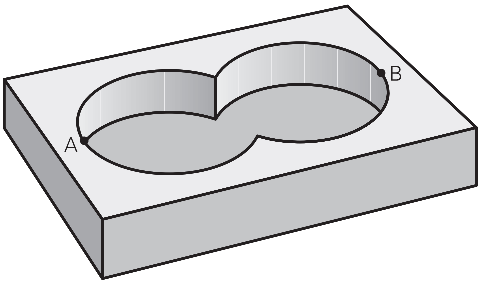
You can use the contour formula function to interlink various contours in a mathematical formula.



- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **Contour formula QC**
- > The control opens the dialog for entering the contour formula.
- ▶ Enter the number for the contour designator **QC**
- ▶ Entering a contour formula

<table>
<tr><th>Help graphic</th><th>Input</th><th>Mathematical function</th><th>Example</th></tr>
<tr><td></td><td>&</td><td>Intersected with</td><td>QC10 = QC1 & QC5</td></tr>
<tr><td></td><td>|</td><td>Joined with</td><td>QC25 = QC7 | QC18</td></tr>
<tr><td></td><td>^</td><td>Joined with, but w/o intersection</td><td>QC12 = QC5 ^ QC25</td></tr>
<tr><td></td><td>\</td><td>Without</td><td>QC25 = QC1 \ QC2</td></tr>
<tr><td></td><td>(</td><td>Opening parenthesis</td><td>QC12 = QC1 & (QC2 | QC3)</td></tr>
<tr><td></td><td>)</td><td>Closing parenthesis</td><td>QC12 = QC1 & (QC2 | QC3)</td></tr>
<tr><td></td><td></td><td>Defining a single contour</td><td>QC12 = QC1</td></tr>
</table>

13.4.5 Superimposed contours



By default, the control considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Subprograms: overlapping pockets

> The following examples are contour description programs that are defined in a contour definition program. The contour definition program is called through the **SEL CONTOUR** function in the actual main program.

Pockets A and B overlap.

The control calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

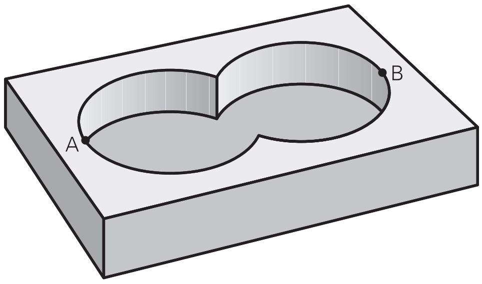
Contour description program 1: pocket A

```
0  BEGIN PGM POCKET MM
1  L  X+10  Y+50 R0
2  CC  X+35  Y+50
3  C  X+10  Y+50 DR-
4  END PGM POCKET MM
```

Contour description program 2: pocket B

```
0  BEGIN PGM POCKET2 MM
1  L  X+90  Y+50 R0
2  CC  X+65  Y+50
3  C  X+90  Y+50 DR-
4  END PGM POCKET2 MM
```

Area of inclusion



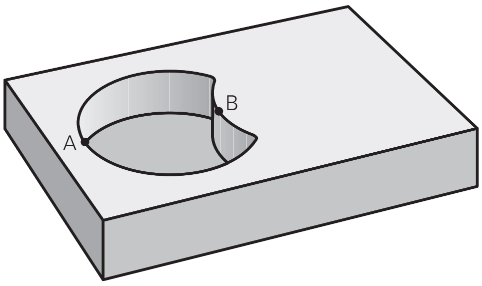
Both areas A and B are to be machined, including the overlapping area:

- Areas A and B must have been programmed in separate NC programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "joined with" function.

Contour definition program:

```
* - ...
21 DECLARE CONTOUR QC1 = "POCKET.H"
22 DECLARE CONTOUR QC2 = "POCKET2.H"
23 QC10 = QC1 | QC2
* - ...
```

Area of exclusion



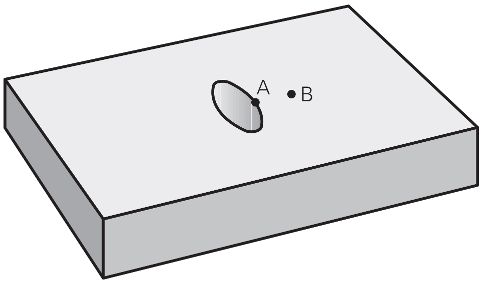
Area A is to be machined without the portion overlapped by B:

- Surfaces A and B must be have been programmed in separate NC programs without radius compensation.
- In the contour formula, the area B is subtracted from the area A with the **without** function.

Contour definition program:

```
* - ...
21 DECLARE CONTOUR QC1 = "POCKET.H"
22 DECLARE CONTOUR QC2 = "POCKET2.H"
23 QC10 = QC1 \ QC2
* - ...
```

Area of intersection



Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- Surfaces A and B must be have been programmed in separate NC programs without radius compensation.
- In the contour formula, the areas A and B are processed with the "intersection with" function.

Contour definition program:

```
* - ...
21 DECLARE CONTOUR QC1 = "POCKET.H"
22 DECLARE CONTOUR QC2 = "POCKET2.H"
23 QC10 = QC1 & QC2
* - ...
```

13.4.6 Machining contours with SL or OCM cycles

> The defined entire contour is machined with the SL cycles or the OCM cycles Page 452.

13.5 Pattern definition with PATTERN DEF

13.5.1 Application

You use the **PATTERN DEF** function to easily define regular machining patterns, which you can call with the **CYCL CALL PAT** function. Just like in cycle definitions, help graphics are available for pattern definition that clearly indicate the input parameters required.

NOTICE

Danger of collision!

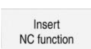
The **PATTERN DEF** function calculates the machining coordinates in the **X** and **Y** axes. For all tool axes apart from **Z** there is a danger of collision in the following operation!

- Use **PATTERN DEF** only in connection with the tool axis **Z**

Possible setting	Definition	Further information
POS1	Point Definition of up to any 9 machining positions	Page 367
ROW1	Row Definition of a single row, straight or rotated	Page 368
PAT1	Pattern Definition of a single pattern, straight, rotated or distorted	Page 369
FRAME1	Frame Definition of a single frame, straight, rotated or distorted	Page 371
CIRC1	Circle Definition of a full circle	Page 373
PITCH-CIRC1	Pitch circle Definition of a pitch circle	Page 374

13.5.2 Entering PATTERN DEF

Proceed as follows:



- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **PATTERN DEF**
- > The control opens the dialog for entering **PATTERN DEF**.
- Select the desired machining pattern, e.g. **CIRC1** for a full circle
- Enter the required definitions
- Define machining cycle, e.g. cycle **200 DRILLING**
- Call cycle with **CYCL CALL PAT**

13.5.3 Using PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the **CYCL CALL PAT** function.

Further information: "Programming a machining cycle", Page 125

The control performs the most recently defined machining cycle on the machining pattern you defined.

Program structure: Machining with PATTERN DEF

```
0 BEGIN SL 2 MM
...
11 PATTERN DEF POS1 (X+25 Y+33.5 Z+0) POS2 (X+15 IY+6.5 Z+0)
12 CYCL DEF 200 DRILLING
...
13 CYCL CALL PAT
```

Notes

Programming note

- Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function with **Q345=1**. Then, between the holes, the control always positions the tool to the 2nd set-up clearance that was defined in the cycle.

Operating notes:

- A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.

  Further information: "Selecting the point table in the NC program with SEL PATTERN", Page 347
- The control retracts the tool to the clearance height between the starting points. Depending on which is greater, the control uses either the tool axis position from the cycle call or the value from cycle parameter **Q204** as the clearance height.
- If the coordinate surface in **PATTERN DEF** is larger than in the cycle, the set-up clearance and the 2nd set-up clearance reference the coordinate surface in **PATTERN DEF**.
- You can use the mid-program startup function to select any point at which you want to start or continue machining.

  Further information: "Block scan for mid-program startup", Page 1786

13.5.4 Defining individual machining positions

Programming and operating notes:

- You can enter up to 9 machining positions. Confirm each entry with the **ENT** key.
- **POS1** must be programmed with absolute coordinates. **POS2** to **POS9** can be programmed as absolute or incremental values.
- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic	Parameter
	POS1: **X coord. of machining position** Enter the X coordinate as an absolute value. Input: **-999999999...+999999999**
	POS1: **Y coord. of machining position** Enter the Y coordinate as an absolute value. Input: **-999999999...+999999999**
	POS1: **Coordinate of workpiece surface** Enter the Z coordinate as an absolute value at which machining starts. Input: **-999999999...+999999999**
	POS2: **X coord. of machining position** Enter the X coordinate as an incremental or absolute value. Input: **-999999999...+999999999**
	POS2: **Y coord. of machining position** Enter the Y coordinate as an incremental or absolute value. Input: **-999999999...+999999999**
	POS2: **Coordinate of workpiece surface** Enter the Z coordinate as an incremental or absolute value. Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
POS1( X+25 Y+33.5 Z+0 ) ~
POS2(  X+15 IY+6.5  Z+0 )
```

13.5.5 Defining a single row

Programming and operating note:

- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic	Parameter
	Starting point in X Coordinate of the starting point of the row in the X axis. The value has an absolute effect. Input: **-99999.9999999...+99999.9999999**
	Starting point in Y Coordinate of the starting point of the row in the Y axis. The value has an absolute effect. Input: **-99999.9999999...+99999.9999999**
	Spacing of machining positions Distance (incremental) between the machining positions. Enter a positive or negative value Input: **-999999999...+999999999**
	Number of operations Total number of machining operations Input: **0...999**
	Rot. position of entire pattern Angle of rotation around the entered starting point. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). Enter a positive or negative absolute value Input: **-360.000...+360.000**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
ROW1( X+25 Y+33.5 D+8 NUM5 ROT+0 Z+0 )
```

13.5.6 Defining an individual pattern

Programming and operating notes:

- The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.
- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic	Parameter
	Starting point in X Absolute coordinate of the pattern starting point in the X axis Input: **-999999999...+999999999**
	Starting point in Y Absolute coordinate of the pattern starting point in the Y axis Input: **-999999999...+999999999**
	Spacing of machining positions X Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Spacing of machining positions Y Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Number of columns Total number of columns in the pattern Input: **0...999**
	Number of rows Total number of rows in the pattern Input: **0...999**
	Rot. position of entire pattern Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). Enter a positive or negative absolute value Input: **-360.000...+360.000**
	Rotary pos. ref. ax. Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value Input: **-360.000...+360.000**

Help graphic	Parameter
	Rotary pos. minor ax. Angle of rotation around which only the secondary axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value Input: **-360.000...+360.000**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts. Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
PAT1( X+25 Y+33.5 DX+8 DY+10 NUMX5 NUMY4 ROT+0 ROTX+0 ROTY+0 Z+0 )
```

13.5.7 Defining an individual frame

Programming and operating notes:

- The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.
- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic	Parameter
	Starting point in X Absolute coordinate of the frame starting point in the X axis Input: **-999999999...+999999999**
	Starting point in Y Absolute coordinate of the frame starting point in the Y axis Input: **-999999999...+999999999**
	Spacing of machining positions X Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Spacing of machining positions Y Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value Input: **-999999999...+999999999**
	Number of columns Total number of columns in the pattern Input: **0...999**
	Number of rows Total number of rows in the pattern Input: **0...999**
	Rot. position of entire pattern Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). Enter a positive or negative absolute value Input: **-360.000...+360.000**
	Rotary pos. ref. ax. Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value. Input: **-360.000...+360.000**

Help graphic	Parameter
	Rotary pos. minor ax. Angle of rotation around which only the secondary axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value. Input: **-360.000...+360.000**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
FRAME1( X+25 Y+33.5 DX+8 DY+10 NUMX5 NUMY4 ROT+0 ROTX+0 ROTY+0 Z+0 )
```

13.5.8 Defining a full circle

Programming and operating notes:

- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic	Parameter
	Bolt-hole circle center X Absolute coordinate of the circle center point in the X axis Input: **-999999999...+999999999**
	Bolt-hole circle center Y Absolute coordinate of the circle center point in the Y axis Input: **-999999999...+999999999**
	Bolt-hole circle diameter Diameter of the bolt hole circle Input: **0...999999999**
	Starting angle Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value Input: **-360.000...+360.000**
	Number of operations Total number of machining positions on the circle Input: **0...999**
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts. Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
CIRC1( X+25 Y+33 D80 START+45 NUM8 Z+0 )
```

13.5.9 Defining a pitch circle

Programming and operating notes:

- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic	Parameter
	Bolt-hole circle center X Absolute coordinate of the circle center point in the X axis Input: **-999999999...+999999999**
	Bolt-hole circle center Y Absolute coordinate of the circle center point in the Y axis Input: **-999999999...+999999999**
	Bolt-hole circle diameter Diameter of the bolt hole circle Input: **0...999999999**
	Starting angle Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g. X for tool axis Z). You can enter a positive or negative value Input: **-360.000...+360.000**
	Stepping angle/Stopping angle Incremental polar angle between two machining positions. You can enter a positive or negative value. As an alternative you can enter the Stopping angle (switch via the selection possibility on the action bar or in the form) Input: **-360.000...+360.000**
	Number of operations Total number of machining positions on the circle Input: **0...999**
	Coordinate of workpiece surface Enter the Z coordinate at which machining starts. Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
PITCHCIRC1( X+25 Y+33 D80 START+45 STEP+30 NUM8 Z+0 )
```

13.5.10 Example: Using cycles in connection with PATTERN DEF

The drill hole coordinates are stored in the PATTERN DEF POS pattern definition. The control calls the drill hole coordinates with CYCL CALL PAT.

The tool radii have been selected in such a way that all work steps can be seen in the test graphics.

Program sequence

- Centering (tool radius 4)
- **GLOBAL DEF 125 POSITIONING**: This function is used for CYCL CALL PAT and positions the tool at the 2nd set-up clearance between the points. This function remains active until M30 is executed.
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)

Further information: "Technology-independent cycles", Page 433 and "Cycles for milling"

0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	; Tool call: centering tool (tool radius 4)
4 L Z+50 R0 FMAX	; Move tool to clearance height
5 PATTERN DEF ~	
POS1(X+10 Y+10 Z+0) ~	
POS2(X+40 Y+30 Z+0) ~	
POS3(X+20 Y+55 Z+0) ~	
POS4(X+10 Y+90 Z+0) ~	
POS5(X+90 Y+90 Z+0) ~	
POS6(X+80 Y+65 Z+0) ~	
POS7(X+80 Y+30 Z+0) ~	
POS8(X+90 Y+10 Z+0)	
6 CYCL DEF 240 CENTERING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q343=+0 ;SELECT DIA./DEPTH ~	
Q201=-2 ;DEPTH ~	
Q344=-10 ;DIAMETER ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE ~	
Q342=+0 ;ROUGHING DIAMETER ~	
Q253=+750 ;F PRE-POSITIONING	
7 GLOBAL DEF 125 POSITIONING ~	
Q345=+1 ;SELECT POS. HEIGHT	
8 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
9 L Z+100 R0 FMAX	; Retract the tool
10 TOOL CALL 227 Z S5000	; Tool call: drill (radius 2.4)

11 L X+50 R0 F5000	; Move tool to clearance height
12 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-25 ;DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;DEPTH REFERENCE	
13 CYCL CALL PAT F500 M3	; Cycle call in connection with the point pattern
14 L Z+100 R0 FMAX	; Retract the tool
15 TOOL CALL 263 Z S200	; Tool call: tap (radius 3)
16 L Z+100 R0 FMAX	; Move tool to clearance height
17 CYCL DEF 206 TAPPING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-25 ;DEPTH OF THREAD ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE	
18 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
19 L Z+100 R0 FMAX	; Retract the tool, end program
20 M30	
21 END PGM 1 MM	

13.6 Cycles for pattern definition

13.6.1 Overview

The control provides three cycles for machining point patterns:

Cycle		Call	Further information
220	**POLAR PATTERN** ■ Defining a circular pattern ■ Full circle or pitch circle ■ Input of start and end angles	**DEF**-active	Page 378
221	**CARTESIAN PATTERN** ■ Defining a linear pattern ■ Input of an angle of rotation	**DEF**-active	Page 381
224	**DATAMATRIX CODE PATTERN** ■ Converting text to a DataMatrix code to be used as a point pattern ■ Input of position and size	**DEF**-active	Page 385

13.6.2 Cycle 220 POLAR PATTERN

Application

This cycle enables you to define a point pattern as a full or pitch circle. It can be used for a previously defined machining cycle.

Cycle sequence

1 The control moves the tool at rapid traverse from its current position to the starting point for the first machining operation.

   Sequence:
   - Move to 2nd set-up clearance (spindle axis)
   - Approach the starting point in the working plane
   - Move to set-up clearance above the workpiece surface (spindle axis)
2 From this position, the control executes the last defined fixed machining cycle
3 The tool then approaches the starting point for the next machining operation on a straight line or a circular arc. The tool stops at the set-up clearance (or the 2nd set-up clearance)
4 This procedure (steps 1 to 3) will be repeated until all machining operations have been completed

> If you run this cycle in **Program Run / Single Block** mode, the control stops between the individual points of a point pattern.

Notes

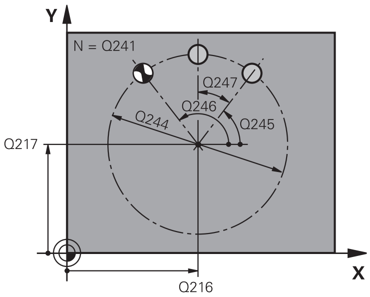
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **220** is DEF-active. In addition, Cycle **220** automatically calls the last defined machining cycle.

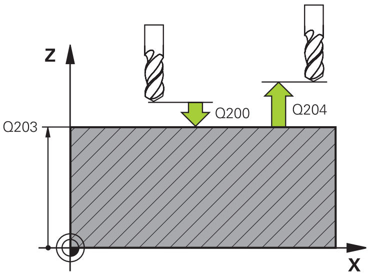
Note on programming

- If you combine one of the machining cycles **200** to **209** or **251** to **267** with Cycle **220** or Cycle **221**, the set-up clearance, the workpiece surface, and the 2nd set-up clearance from Cycle **220** or **221** are effective. This applies within the NC program until the affected parameters are overwritten again.

  Example: If Cycle **200** is defined in an NC program with **Q203**=0 and you then program Cycle **220** with **Q203**=-5, then the subsequent calls with **CYCL CALL** and **M99** will use **Q203**=-5. Cycles **220** and **221** overwrite the above-mentioned parameters of **CALL**-active machining cycles (if the same input parameters have been programmed in both cycles).

Cycle parameters

Help graphic	Parameter
	**Q216 Center in 1st axis?** Pitch circle center in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q217 Center in 2nd axis? Pitch circle center in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q244 Pitch circle diameter? Diameter of circle Input: **0...99999.9999**
	Q245 Starting angle? Angle between the main axis of the working plane and the starting point for the first machining operation on the pitch circle. The value has an absolute effect. Input: **-360.000...+360.000**
	Q246 Stopping angle? Angle between the main axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you specify a stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. The value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the control will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the control will not take the stopping angle into account. The sign for the angle step determines the working direction (negative = clockwise). This value has an incremental effect. Input: **-360.000...+360.000**
	Q241 Number of repetitions? Number of machining operations on a pitch circle Input: **1...99999**

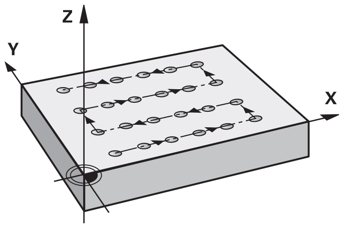
Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the tool moves between machining processes: **0**: Move to the set-up clearance between operations **1**: Move to the 2nd set-up clearance between operations Input: **0**, **1**
	Q365 Type of traverse? Line=0/arc=1 Specify how the tool moves between machining processes: **0**: Move between operations on a straight line **1**: Move between operations on the pitch circle Input: **0**, **1**

Example

```
11 CYCL DEF 220 POLAR PATTERN ~
    Q216=+50          ;CENTER IN 1ST AXIS ~
    Q217=+50          ;CENTER IN 2ND AXIS ~
    Q244=+60          ;PITCH CIRCLE DIAMETR ~
    Q245=+0           ;STARTING ANGLE ~
    Q246=+360         ;STOPPING ANGLE ~
    Q247=+0           ;STEPPING ANGLE ~
    Q241=+8           ;NR OF REPETITIONS ~
    Q200=+2           ;SET-UP CLEARANCE ~
    Q203=+0           ;SURFACE COORDINATE ~
    Q204=+50          ;2ND SET-UP CLEARANCE ~
    Q301=+1           ;MOVE TO CLEARANCE ~
    Q365=+0           ;TYPE OF TRAVERSE
12 CYCL CALL
```

13.6.3 Cycle 221 CARTESIAN PATTERN

Application



This cycle enables you to define a point pattern as lines. It can be used for a previously defined machining cycle.

Cycle sequence

1 The control automatically moves the tool from its current position to the starting point for the first machining operation

  Sequence:
  - Move to 2nd set-up clearance (spindle axis)
  - Approach the starting point in the working plane
  - Move to set-up clearance above the workpiece surface (spindle axis)

2 From this position, the control executes the last defined fixed machining cycle
3 Then, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis. The tool stops at the set-up clearance (or the 2nd set-up clearance)
4 This procedure (steps 1 to 3) will be repeated until all machining operations from the first line have been completed. The tool is located above the last point of the first line
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation
6 From this position, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis.
7 This procedure (step 6) will be repeated until all machining operations of the second line have been completed
8 The tool then moves to the starting point of the next line
9 All subsequent lines are machined in a reciprocating movement.

i	If you run this cycle in **Program Run / Single Block** mode, the control stops between the individual points of a point pattern.

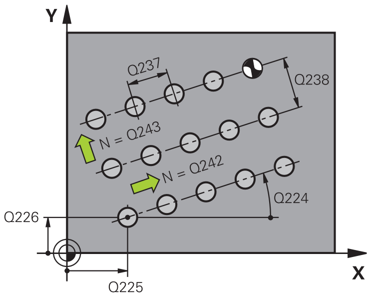
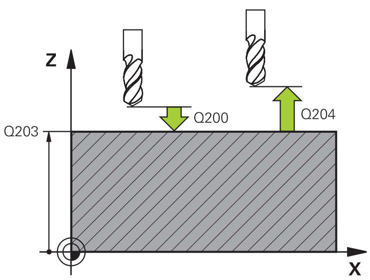
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **221** is DEF-active. In addition, Cycle **221** automatically calls the last defined machining cycle.

Notes on programming

- If you combine one of the machining cycles **200** to **209** or **251** to **267** with Cycle **221**, the set-up clearance, the workpiece surface, the 2nd safety clearance, and the rotary position from Cycle **221** are effective.
- Slot position 0 is not allowed if you use Cycle **254** in combination with Cycle **221**.

Cycle parameters

Help graphic	Parameter
	**Q225 Starting point in 1st axis?** Coordinate of starting point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q226 Starting point in 2nd axis? Coordinate of starting point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q237 Spacing in 1st axis? Spacing between the individual points on a line. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q238 Spacing in 2nd axis? Spacing between the individual lines. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q242 Number of columns? Number of machining operations on a line Input: **0...99999**
	Q243 Number of lines? Number of lines Input: **0...99999**
	Q224 Angle of rotation? Angle by which the entire pattern is rotated. The center of rotation lies in the starting point. The value has an absolute effect. Input: **-360.000...+360.000**
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the tool moves between machining processes: **0**: Move to the set-up clearance between operations **1**: Move to the 2nd set-up clearance between operations Input: **0**, **1**

Example

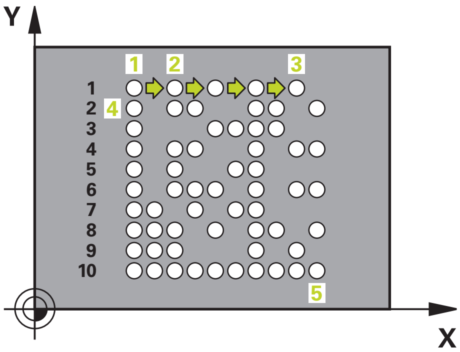
```
11 CYCL DEF 221 CARTESIAN PATTERN ~
    Q225=+15                ;STARTNG PNT 1ST AXIS ~
    Q226=+15                ;STARTNG PNT 2ND AXIS ~
    Q237=+10                ;SPACING IN 1ST AXIS ~
    Q238=+8                 ;SPACING IN 2ND AXIS ~
    Q242=+6                 ;NUMBER OF COLUMNS ~
    Q243=+4                 ;NUMBER OF LINES ~
    Q224=+15                ;ANGLE OF ROTATION ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q301=+1                 ;MOVE TO CLEARANCE
12 CYCL CALL
```

13.6.4 Cycle 224 DATAMATRIX CODE PATTERN

Application

Use Cycle **224 DATAMATRIX CODE PATTERN** to convert text to a so-called DataMatrix code. This code will be used as a point pattern for a previously defined fixed cycle.

Cycle sequence



1 The control automatically moves the tool from its current position to the programmed starting point. This point is always located in the lower left corner.
   Sequence:
   - Move to 2nd set-up clearance (spindle axis)
   - Approach the starting point in the working plane
   - Move to the **Safety clearance** above the workpiece surface (spindle axis)
2 Then, the control moves the tool in the positive direction of the secondary axis to the first point **1** in the first row
3 From this position, the control executes the last defined fixed machining cycle
4 Then, the control moves the tool in the positive direction of the principal axis to point **2** for the next operation.
5 This procedure will be repeated until all machining operations in the first row have been completed. The tool is located above the last point **3** of the first row
6 Then, the control moves the tool in the negative direction of the principal and secondary axes to the first point **4** of the next row
7 Then, the next points are machined
8 These steps are repeated until the entire DataMatrix code has been completed. Machining stops in the lower right corner **5**
9 Finally, the control retracts the tool to the programmed 2nd set-up clearance

Notes

NOTICE

Danger of collision!

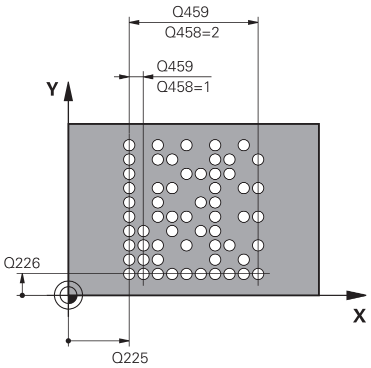
If you combine Cycle **224** with one of the machining cycles, the **Safety clearance**, coordinate surface and 2nd set-up clearance that you defined in Cycle **224** will be effective for the selected machining cycle.

- Check the machining sequence using a graphic simulation
- Carefully test the NC program or program section in the **SINGLE BLOCK** mode of the **Program run** operating mode.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **224** is DEF-active. In addition, Cycle **224** automatically calls the last defined machining cycle.
- The control uses the special character **%** for special functions. If you want to use this character in a DataMatrix code, enter it twice in the text, e.g. **%%**).

Cycle parameters

Help graphic	Parameter



Q225 Starting point in 1st axis?

Coordinate in the lower left corner of the code in the main axis. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q226 Starting point in 2nd axis?

Coordinate in the lower left corner of the code in the secondary axis. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

QS501 Text input?

Enter the text to be converted within quotation marks. Variables can be assigned.

Further information: "Outputting variable texts in DataMatrix codes", Page 388

Input: Max. **255** characters

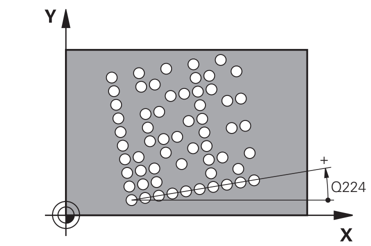
Q458 Cell size/Pattern size(1/2)?

Specify how the DataMatrix code is described in **Q459**:

1: Distance between cells

2: Pattern size

Input: **1**, **2**



Q459 Size for pattern?

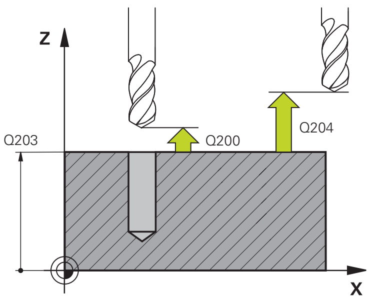
Definition of the distance between cells or the pattern size:

If **Q458=1**: Distance between the first and second cell (between cell centers)

If **Q458=2**: Distance between the first and last cell (between cell centers)

This value has an incremental effect.

Input: **0...99999.9999**



Q224 Angle of rotation?

Angle by which the entire pattern is rotated. The center of rotation lies in the starting point. The value has an absolute effect.

Input: **-360.000...+360.000**

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 CYCL DEF 224 DATAMATRIX CODE PATTERN ~
    Q225=+0              ;STARTNG PNT 1ST AXIS ~
    Q226=+0              ;STARTNG PNT 2ND AXIS ~
    QS501=""             ;TEXT ~
    Q458=+1              ;SIZE SELECTION ~
    Q459=+1              ;SIZE ~
    Q224=+0              ;ANGLE OF ROTATION ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE
12 CYCL CALL
```

Outputting variable texts in DataMatrix codes

In addition to specified characters you can also output certain variables in DataMatrix codes. Precede the variable with **%**.

You can use the following variable texts in Cycle **224 DATAMATRIX CODE PATTERN**:

- Date and time
- Names and paths of NC programs
- Count values

Date and time

You can convert the current date, the current time, or the current calendar week into a DataMatrix code. Enter the value **%time<x>** in cycle parameter **QS501**. **<x>** defines the format, e.g. 08 for DD.MM.YYYY.

> Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9, e.g. **%time08**.

The following formats are available:

Input	Format
%time00	DD.MM.YYYY hh:mm:ss
%time01	D.MM.YYYY h:mm:ss
%time02	D.MM.YYYY h:mm
%time03	D.MM.YY h:mm
%time04	YYYY-MM-DD hh:mm:ss
%time05	YYYY-MM-DD hh:mm
%time06	YYYY-MM-DD h:mm
%time07	YY-MM-DD h:mm
%time08	DD.MM.YYYY
%time09	D.MM.YYYY
%time10	D.MM.YY
%time11	YYYY-MM-DD
%time12	YY-MM-DD
%time13	hh:mm:ss
%time14	h:mm:ss
%time15	h:mm
%time99	Calendar week

Names and paths of NC programs

You can convert the name or path of the active or called NC program into a DataMatrix code. Enter the value **%main<x>** or **%prog<x>** in cycle parameter **QS501**.

The following formats are available:

Input	Meaning	Example
%main0	Full path of the active NC program	**TNC:\MILL.h**
%main1	Directory path of the active NC program	**TNC:**
%main2	Name of the active NC program	**MILL**
%main3	File type of the active NC program	**.H**
%prog0	Full path of the called NC program	**TNC:\HOUSE.h**
%prog1	Directory path of the called NC program	**TNC:**
%prog2	Name of the called NC program	**HOUSE**
%prog3	File type of the called NC program	**.H**

Count values

You can convert the current count value into a DataMatrix code. The control displays the current count value in **Program Run** on the**PGM** tab of the **Status** workspace.

Enter the value **%count<x>** in cycle parameter **QS501**.

The number after **%count** indicates how many digits the DataMatrix code contains. The maximum is nine digits.

Example:

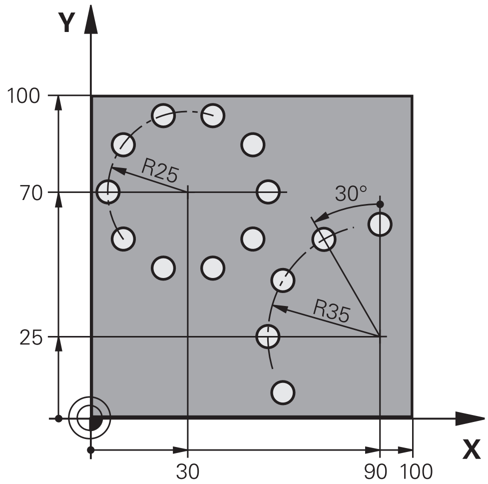
- Programming: **%count9**
- Current count value: 3
- Result: 000000003

Operating information

- In the Simulation, the control only simulates the count value you define directly in the NC program. The count value from the **Status** workspace in the **Program Run**operating mode is ignored.

13.6.5 Programming Examples

Example: Polar hole patterns



0 BEGIN PGM 200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 200 Z S3500	; Tool call
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-15 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+4 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q211=+0.25 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;DEPTH REFERENCE	
6 CYCL DEF 220 POLAR PATTERN ~	
Q216=+30 ;CENTER IN 1ST AXIS ~	
Q217=+70 ;CENTER IN 2ND AXIS ~	
Q244=+50 ;PITCH CIRCLE DIAMETR ~	
Q245=+0 ;STARTING ANGLE ~	
Q246=+360 ;STOPPING ANGLE ~	
Q247=+0 ;STEPPING ANGLE ~	
Q241=+10 ;NR OF REPETITIONS ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+100 ;2ND SET-UP CLEARANCE ~	
Q301=+1 ;MOVE TO CLEARANCE ~	
Q365=+0 ;TYPE OF TRAVERSE	

7 CYCL DEF 220 POLAR PATTERN ~	
Q216=+90 ;CENTER IN 1ST AXIS ~	
Q217=+25 ;CENTER IN 2ND AXIS ~	
Q244=+70 ;PITCH CIRCLE DIAMETR ~	
Q245=+90 ;STARTING ANGLE ~	
Q246=+360 ;STOPPING ANGLE ~	
Q247=+30 ;STEPPING ANGLE ~	
Q241=+5 ;NR OF REPETITIONS ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+100 ;2ND SET-UP CLEARANCE ~	
Q301=+1 ;MOVE TO CLEARANCE ~	
Q365=+0 ;TYPE OF TRAVERSE	
8 L Z+100 R0 FMAX	; Retract the tool
9 M30	; End of program
10 END PGM 200 MM	

13.7 OCM cycles for pattern definition

13.7.1 Overview

OCM figures

Cycle	Call	Further information
1271 OCM RECTANGLE (option 167) ■ Definition of a rectangle ■ Input of the side lengths ■ Definition of the corners	**DEF**-active	Page 395
1272 OCM CIRCLE (option 167) ■ Definition of a circle ■ Input of the circle diameter	**DEF**-active	Page 398
1273 OCM SLOT / RIDGE (option 167) ■ Definition of a slot or ridge ■ Input of the width and the length	**DEF**-active	Page 400
1278 OCM POLYGON (option 167) ■ Definition of a polygon ■ Input of the reference circle ■ Definition of the corners	**DEF**-active	Page 403
1281 OCM RECTANGLE BOUNDARY (option 167) ■ Definition of a bounding rectangle	**DEF**-active	Page 406
1282 OCM CIRCLE BOUNDARY (option 167) ■ Definition of a bounding circle	**DEF**-active	Page 407

13.7.2 Fundamentals

The control provides cycles for frequently used figures. You can program these figures as pockets, islands, or boundaries.

These figure cycles offer the following benefits:

- You can conveniently program the figures and machining data without the need to program an individual path contour.
- Frequently needed figures can be reused.
- If you want to program an island or an open pocket, the control provides you with more cycles for defining the figure boundary.
- The Boundary figure type enables you to face-mill your figure.

With a figure, you can redefine the OCM contour data and cancel the definition of a previously defined Cycle **271 OCM CONTOUR DATA** or of a figure boundary.

For defining figures, the control provides the following cycles:

- **1271 OCM RECTANGLE**, see Page 395
- **1272 OCM CIRCLE**, see Page 398
- **1273 OCM SLOT / RIDGE**, see Page 400
- **1278 OCM POLYGON**, see Page 403

For defining figure boundaries, the control provides the following cycles:

- **1281 OCM RECTANGLE BOUNDARY**, see Page 406
- **1282 OCM CIRCLE BOUNDARY**, see Page 407

Tolerances

The control allows you to store tolerances in the following cycles and cycle parameters:

Cycle number	Parameter
1271 OCM RECTANGLE	**Q218 FIRST SIDE LENGTH**, **Q219 2ND SIDE LENGTH**
1272 OCM CIRCLE	**Q223 CIRCLE DIAMETER**
1273 OCM SLOT / RIDGE	**Q219 SLOT WIDTH**, **Q218 SLOT LENGTH**
1278 OCM POLYGON	**Q571 REF-CIRCLE DIAMETER**

You can define the following tolerances:

Tolerances	Example	Manufacturing dimension
Dimensions	10+0.01-0.015	9.9975
DIN EN ISO 286-2	10H7	10.0075
DIN ISO 2768-1	10m	10.0000

> Pay attention to capitalization when entering tolerances.

Proceed as follows:

- Start the cycle definition
- Define the cycle parameters
- Select **TEXT** in the action bar
- Enter a nominal dimension including tolerance

> If you program an incorrect tolerance, the control interrupts machining with an error message.

13.7.3 Cycle 1271 OCM RECTANGLE (option 167)

Application

Use the figure cycle **1271 OCM RECTANGLE** to program a rectangle. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program tolerances for the lengths.

If you work with Cycle **1271**, program the following:

- Cycle **1271 OCM RECTANGLE**
  - If you program **Q650**=1 (figure type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

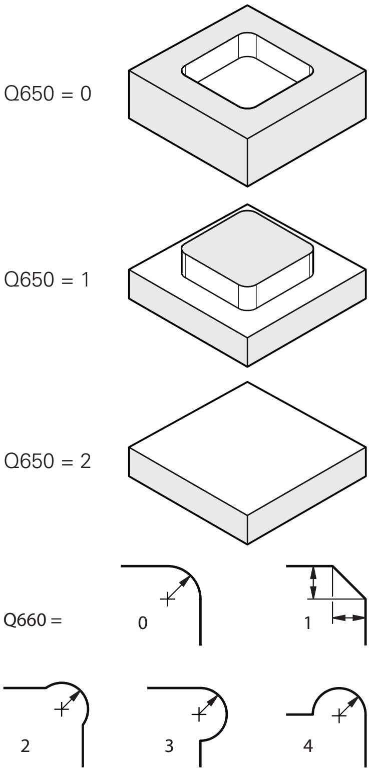
Notes

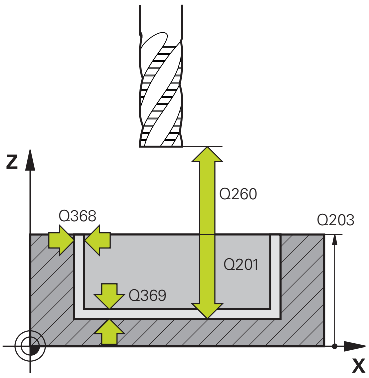
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1271** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1271** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

Cycle parameters

Help graphic	Parameter
 	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q218 First side length? Length of the first side of the figure, parallel to the main axis. This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 394 Input: **0...99999.9999**
	Q219 Second side length? Length of the 2nd side of the figure, parallel to the secondary axis. This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 394 Input: **0...99999.9999**
	Q660 Type of corners? Geometry of the corners: **0**: Radius **1**: Chamfer **2**: Milling corners in the main and secondary axis directions **3**: Milling corners in the main axis direction **4**: Milling corners in the secondary axis direction Input: **0**, **1**, **2**, **3**, **4**
	Q220 Corner radius? Radius or chamfer of the corner of the figure Input: **0...99999.9999**
	Q367 Position of pocket (0/1/2/3/4)? Position of the figure relative to the position of the tool when the cycle is called: **0**: Tool position = Center of figure **1**: Tool position = Lower left corner **2**: Tool position = Lower right corner **3**: Tool position = Upper right corner **4**: Tool position = Upper left corner Input: **0**, **1**, **2**, **3**, **4**
	Q224 Angle of rotation? Angle by which the figure is rotated. The center of rotation is at the center of the figure. The value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	**Q203 Workpiece surface coordinate?** Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners? The inside radii of the contour are calculated based on the tool radius plus the product of the tool radius times **Q578**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1271 OCM RECTANGLE ~
    Q650=+1                ;FIGURE TYPE ~
    Q218=+60               ;FIRST SIDE LENGTH ~
    Q219=+40               ;2ND SIDE LENGTH ~
    Q660=+0                ;CORNER TYPE ~
    Q220=+0                ;CORNER RADIUS ~
    Q367=+0                ;POCKET POSITION ~
    Q224=+0                ;ANGLE OF ROTATION ~
    Q203=+0                ;SURFACE COORDINATE ~
    Q201=-10               ;DEPTH ~
    Q368=+0                ;ALLOWANCE FOR SIDE ~
    Q369=+0                ;ALLOWANCE FOR FLOOR ~
    Q260=+50               ;CLEARANCE HEIGHT ~
    Q578=+0.2              ;INSIDE CORNER FACTOR
```

13.7.4 Cycle 1272 OCM CIRCLE (option 167)

Application

Use figure cycle **1272 OCM CIRCLE** to program a circle. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program a tolerance for the diameter.

If you work with Cycle **1272**, program the following:

- Cycle **1272 OCM CIRCLE**
  - If you program **Q650**=1 (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

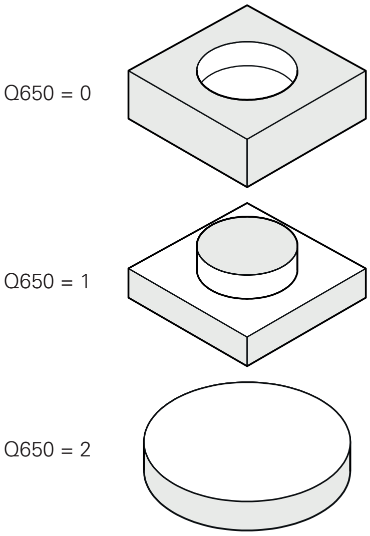
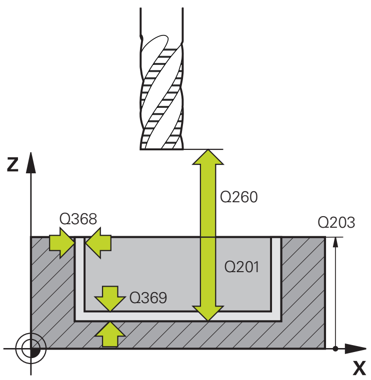
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1272** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1272** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

Cycle parameters

Help graphic	Parameter
	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q223 Circle diameter? Diameter of the finished circle. You can program a tolerance if needed. **Further information:** "Tolerances", Page 394 Input: **0...99999.9999**
	Q367 Position of pocket (0/1/2/3/4)? Position of the figure relative to the position of the tool during the cycle call: **0**: Tool pos. = Center of figure **1**: Tool pos. = Quadrant transition at 90° **2**: Tool pos. = Quadrant transition at 0° **3**: Tool pos. = Quadrant transition at 270° **4**: Tool pos. = Quadrant transition at 180° Input: **0**, **1**, **2**, **3**, **4**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	**Q201 Depth?** Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q578 Radius factor on inside corners? The minimum radius of a circular pocket results from the tool radius plus the product of tool radius and **Q578**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1272 OCM CIRCLE ~
    Q650=+0                    ;FIGURE TYPE ~
    Q223=+50                   ;CIRCLE DIAMETER ~
    Q367=+0                    ;POCKET POSITION ~
    Q203=+0                    ;SURFACE COORDINATE ~
    Q201=-20                   ;DEPTH ~
    Q368=+0                    ;ALLOWANCE FOR SIDE ~
    Q369=+0                    ;ALLOWANCE FOR FLOOR ~
    Q260=+100                  ;CLEARANCE HEIGHT ~
    Q578=+0.2                  ;INSIDE CORNER FACTOR
```

13.7.5 Cycle 1273 OCM SLOT / RIDGE (option 167)

Application

Use figure cycle **1273 OCM SLOT / RIDGE** to program a slot or a ridge. This figure cycle also allows you to program a boundary for face milling. In addition, you can program a tolerance for the width and the length.

If you work with Cycle **1273**, program the following:

- Cycle **1273 OCM SLOT / RIDGE**
  - If you program **Q650**=1 (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

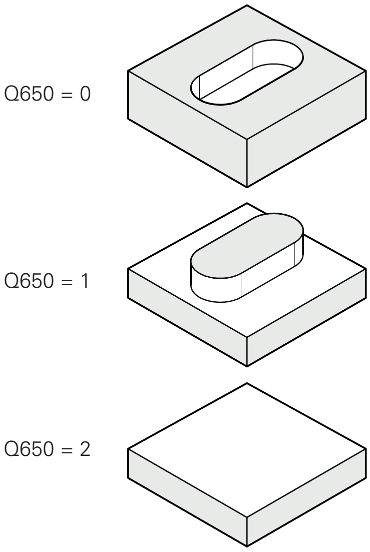
Notes

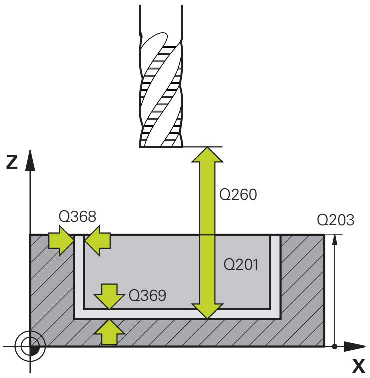
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1273** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1273** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

Cycle parameters

Help graphic	Parameter
	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q219 Width of slot? Width of the slot or ridge, parallel to the secondary axis of the working plane. This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 394 Input: **0...99999.9999**
	Q218 Length of slot? Length of the slot or ridge, parallel to the main axis of the working plane. This value has an incremental effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 394 Input: **0...99999.9999**
	Q367 Position of slot (0/1/2/3/4)? Position of the figure relative to the position of the tool when the cycle is called: **0**: Tool position = Center of figure **1**: Tool position = Left end of figure **2**: Tool position = Center of left figure arc **3**: Tool position = Center of right figure arc **4**: Tool position = Right end of figure Input: **0**, **1**, **2**, **3**, **4**
	Q224 Angle of rotation? Angle by which the figure is rotated. The center of rotation is at the center of the figure. The value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	**Q203 Workpiece surface coordinate?** Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners? The minimum radius (slot width) of a slot results from the tool radius plus the product of tool radius and **Q578**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1273 OCM SLOT / RIDGE ~
    Q650=+0          ;FIGURE TYPE ~
    Q219=+10         ;SLOT WIDTH ~
    Q218=+60         ;SLOT LENGTH ~
    Q367=+0          ;SLOT POSITION ~
    Q224=+0          ;ANGLE OF ROTATION ~
    Q203=+0          ;SURFACE COORDINATE ~
    Q201=-20         ;DEPTH ~
    Q368=+0          ;ALLOWANCE FOR SIDE ~
    Q369=+0          ;ALLOWANCE FOR FLOOR ~
    Q260=+100        ;CLEARANCE HEIGHT ~
    Q578=+0.2        ;INSIDE CORNER FACTOR
```

13.7.6 Cycle 1278 OCM POLYGON (option 167)

Application

Use figure cycle **1278 OCM POLYGON** to program a polygon. You can use the figure to machine a pocket, an island, or a boundary by face milling. In addition, you can program a tolerance for the reference diameter.

If you work with Cycle **1278**, program the following:

- Cycle **1278 OCM POLYGON**
  - If you program **Q650**=1 (shape type = island), you need to define a boundary using Cycle **1281 OCM RECTANGLE BOUNDARY** or **1282 OCM CIRCLE BOUNDARY**
- Cycle **272 OCM ROUGHING**
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable
- Cycle **277 OCM CHAMFERING**, if applicable

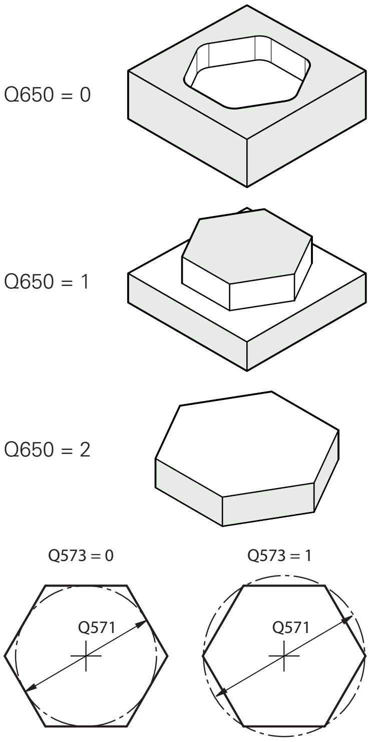
Notes

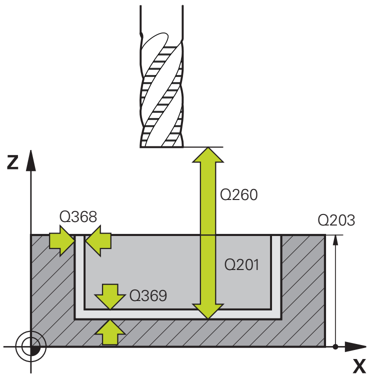
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1278** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **1278** are valid for the OCM machining cycles **272** to **274** and **277**.

Note on programming

- The cycle requires corresponding pre-positioning, depending on the setting in **Q367**.

Cycle parameters

Help graphic	Parameter
	**Q650 Type of figure?** Geometry of the figure: **0**: Pocket **1**: Island **2**: Boundary for face milling Input: **0**, **1**, **2**
	Q573 Inscr.circle/circumcircle (0/1)? Define whether the dimension **Q571** is referenced to the inscribed circle or the circumcircle: **0**: Dimension is referenced to the inscribed circle **1**: Dimension is referenced to the circumcircle Input: **0**, **1**
	Q571 Reference circle diameter? Enter the diameter of the reference circle. Specify in parameter **Q573** whether the diameter entered here is referenced to the inscribed circle or the circumcircle. You can program a tolerance if needed. **Further information:** "Tolerances", Page 394 Input: **0...99999.9999**
	Q572 Number of corners? Enter the number of corners of the polygon. The control will always distribute the corners evenly on the polygon. Input: **3...30**
	Q660 Type of corners? Geometry of the corners: **0**: Radius **1**: Chamfer Input: **0**, **1**
	Q220 Corner radius? Radius or chamfer of the corner of the figure Input: **0...99999.9999**
	Q224 Angle of rotation? Angle by which the figure is rotated. The center of rotation is at the center of the figure. The value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	**Q203 Workpiece surface coordinate?** Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q578 Radius factor on inside corners? The inside radii of the contour are calculated based on the tool radius plus the product of the tool radius times **Q578**. Input: **0.05...0.99**

Example

```
11 CYCL DEF 1278 OCM POLYGON ~
    Q650=+0              ;FIGURE TYPE ~
    Q573=+0              ;REFERENCE CIRCLE ~
    Q571=+50             ;REF-CIRCLE DIAMETER ~
    Q572=+6              ;NUMBER OF CORNERS ~
    Q660=+0              ;CORNER TYPE ~
    Q220=+0              ;CORNER RADIUS ~
    Q224=+0              ;ANGLE OF ROTATION ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q201=-10             ;DEPTH ~
    Q368=+0              ;ALLOWANCE FOR SIDE ~
    Q369=+0              ;ALLOWANCE FOR FLOOR ~
    Q260=+50             ;CLEARANCE HEIGHT ~
    Q578=+0.2            ;INSIDE CORNER FACTOR
```

13.7.7 Cycle 1281 OCM RECTANGLE BOUNDARY (option 167)

Application

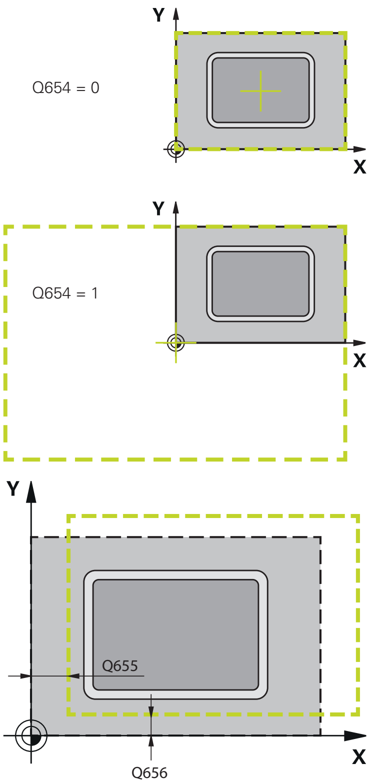
Use Cycle **1281 OCM RECTANGLE BOUNDARY** to program a rectangular bounding frame. This cycle can be used to define the outer boundary of an island or a boundary of an open pocket that was programmed before by using the respective OCM standard figure.

The cycle becomes effective when you program cycle parameter **Q650 FIGURE TYPE** = 0 (pocket) or = 1 (island) within an OCM standard figure cycle.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1281** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The boundary data entered in Cycle **1281** are valid for Cycles **1271** to **1273** and **1278**.

Cycle parameters

Help graphic	Parameter
	**Q651 Length of major axis?** Length of the first side of the boundary, parallel to the main axis Input: **0.001...9999.999**
	Q652 Length of minor axis? Length of the second side of the boundary, parallel to the secondary axis Input: **0.001...9999.999**
	Q654 Position reference for figure? Specify the position reference for the center: **0**: The center of the boundary is referenced to the center of the contour **1**: The center of the boundary is referenced to the datum Input: **0**, **1**
	Q655 Shift in major axis? Shift of the rectangle boundary along the main axis Input: **-999.999...+999.999**
	Q656 Shift in minor axis? Shift of the rectangle boundary along the secondary axis Input: **-999.999...+999.999**

Example

```
11 CYCL DEF 1281 OCM RECTANGLE BOUNDARY ~
    Q651=+50                 ;LENGTH 1 ~
    Q652=+50                 ;LENGTH 2 ~
    Q654=+0                  ;POSITION REFERENCE ~
    Q655=+0                  ;SHIFT 1 ~
    Q656=+0                  ;SHIFT 2
```

13.7.8 Cycle 1282 OCM CIRCLE BOUNDARY (option 167)

Application

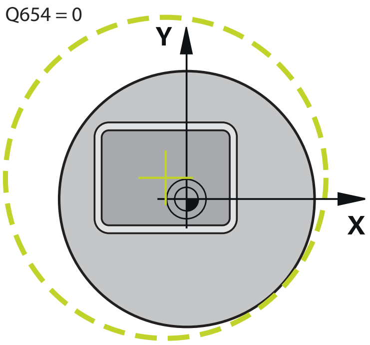
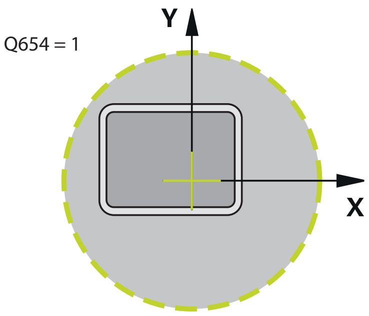
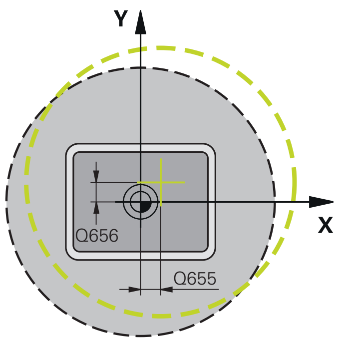
Cycle **1282 OCM CIRCLE BOUNDARY** allows you to program a circular bounding frame. This cycle can be used to define the outer boundary of an island or a boundary of an open pocket that was programmed before by using the respective OCM standard figure.

The cycle becomes effective when you program cycle parameter **Q650 FIGURE TYPE** = **0** (pocket) or = **1** (island) in an OCM standard shape cycle.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1282** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The boundary data entered in Cycle **1282** are valid for Cycles **1271** to **1273** and **1278**.

Cycle parameters

Help graphic	Parameter
	**Q653 Diameter?** Diameter of the circular bounding frame Input: **0.001...9999.999**
	Q654 Position reference for figure? Specify the position reference for the center: **0**: The center of the boundary is referenced to the center of the contour **1**: The center of the boundary is referenced to the datum Input: **0**, **1**
	**Q655 Shift in major axis?** Shift of the rectangle boundary along the main axis Input: **-999.999...+999.999**
	**Q656 Shift in minor axis?** Shift of the rectangle boundary along the secondary axis Input: **-999.999...+999.999**

Example

```
11 CYCL DEF 1282 OCM CIRCLE BOUNDARY ~
    Q653=+50                  ;DIAMETER ~
    Q654=+0                   ;POSITION REFERENCE ~
    Q655=+0                   ;SHIFT 1 ~
    Q656=+0                   ;SHIFT 2
```

13.8 Recessing and undercutting

13.8.1 Recessing and undercutting

Some cycles machine contours that you have written in a subprogram. Further special contour elements are available to you for writing turning contours. In this way you can program recessing and undercutting as complete contour elements with a single NC block.

Recessing and undercutting are always referenced to a previously defined linear contour element.

You can only use the recess and undercut elements GRV and UDC in contour subprograms that have been called by a turning cycle.

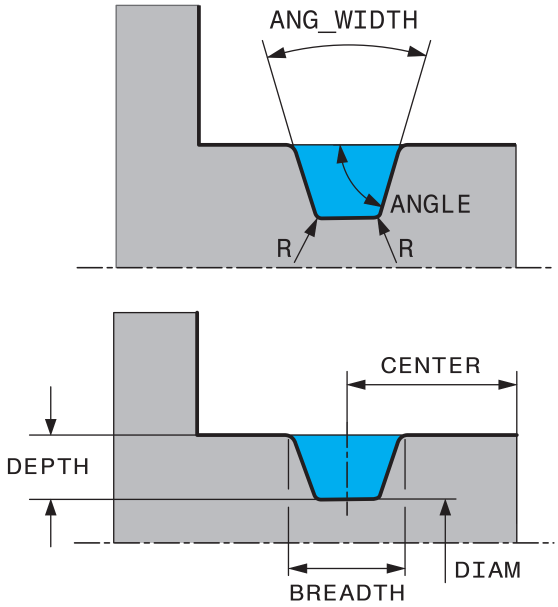
Various input options are available to you for defining undercuts and recesses. Some of these inputs have to be made (mandatory input); others can be skipped (optional input). The mandatory inputs are symbolized as such in the help graphics. In some elements, you can select between two different definitions. The control provides relevant selection possibilities via an action bar.

The control provides various possibilities for programming recesses and undercuts in the **Recess / Undercut** file of the **Insert NC function** window.

Programming recessing

Recessing is the machining of recesses into round parts, usually for accommodation of locking rings and seals, or as lubricating grooves. You can program recessing around the circumference or on the face end of the turned part. You have two separate contour elements for this purpose:

- **GRV RADIAL**: Recess in circumference of component
- **GRV AXIAL**: Recess on face end of component



Input parameters in recessing GRV

Parameter	Meaning	Input
CENTER	Center of recess	Required
R	Corner radius of both inside corners	Optional
DEPTH / **DIAM**	Depth of recess (pay attention to algebraic sign!) /diameter of recess base	Required
BREADTH	Recess width	Required
ANGLE / **ANG_WIDTH**	Flank angle / opening angle between both flanks	Optional
RND / **CHF**	Rounding / chamfer on contour corner near to starting point	Optional
FAR_RND / **FAR_CHF**	Rounding / chamfer on contour corner away from starting point	Optional

> The algebraic sign for the recess depth specifies the machining position (inside/outside machining) of the recess.
> Algebraic signs of recess depth for outside machining:
> - If the contour element is in the negative direction of the Z coordinate, use a negative sign
> - If the contour element is in the positive direction of the Z coordinate, use a positive sign
>
> Algebraic signs of recess depth for inside machining:
> - If the contour element is in the negative direction of the Z coordinate, use a positive sign
> - If the contour element is in the positive direction of the Z coordinate, use a negative sign

Example: Radial recess with depth=5, width=10, pos.= Z-15

```
11 L  X+40  Z+0
12 L  Z-30
13 GRV RADIAL CENTER-15 DEPTH-5 BREADTH10 CHF1 FAR_CHF1
14 L  X+60
```

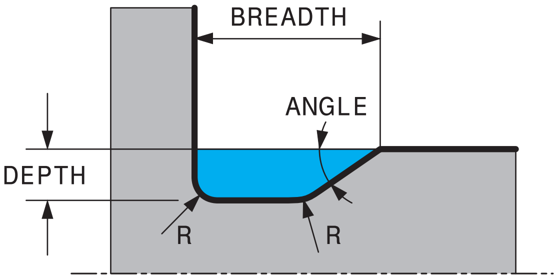
Programming undercutting

Undercutting is usually required for the flush connection of components. In addition, undercutting can help reduce the notch effect at corners. Threads and fits are often machined with an undercut. You have various contour elements for defining the different undercuts:

- **UDC TYPE_E**: Undercut for cylindrical surfaces to be further processed as per DIN 509.
- **UDC TYPE_F**: Undercut for plane surface and cylindrical surface to be further processed as per DIN 509
- **UDC TYPE_H**: Undercut for more rounded transition as per DIN 509
- **UDC TYPE_K**: Undercut in plane surface and cylindrical surface
- **UDC TYPE_U**: Undercut in cylindrical surface
- **UDC THREAD**: Thread undercut as per DIN 76

> The control always interprets undercuts as form elements in the longitudinal direction. No undercuts are possible in the plane direction.

Undercut DIN 509 UDC TYPE _E



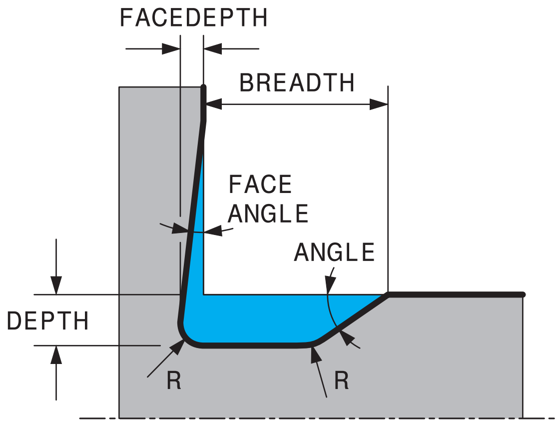
Input parameters in undercut DIN 509 UDC TYPE_E

Parameter	Meaning	Input
R	Corner radius of both inside corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional

Example: Undercut with depth = 2, width = 15

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_E R1 DEPTH2 BREADTH15
14 L  X+60
```

Undercut DIN 509 UDC TYPE_F



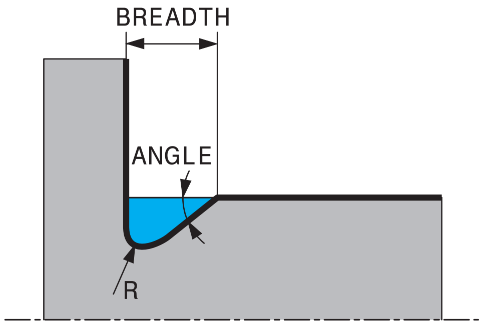
Input parameters in undercut DIN 509 UDC TYPE_F

Parameter	Meaning	Input
R	Corner radius of both inside corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional
FACEDEPTH	Depth of face	Optional
FACEANGLE	Contour angle of face	Optional

Example: Undercut form F with depth = 2, Width = 15, Depth of face = 1

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_F R1 DEPTH2 BREADTH15 FACEDEPTH1
14 L  X+60
```

Undercut DIN 509 UDC TYPE_H



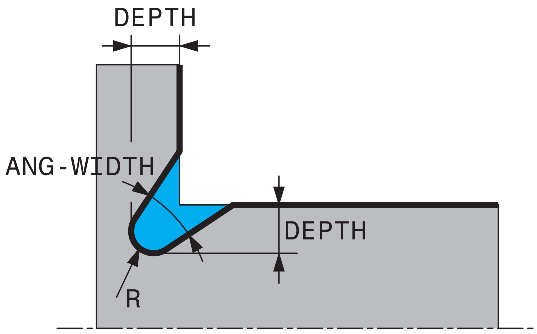
Input parameters in undercut DIN 509 UDC TYPE_H

Parameter	Meaning	Input
R	Corner radius of both inside corners	Required
BREADTH	Width of undercut	Required
ANGLE	Undercut angle	Required

Example: Undercut form H with depth = 2, width = 15, angle = 10°

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_H R1 BREADTH10 ANGLE10
14 L  X+60
```

Undercut UDC TYPE_K



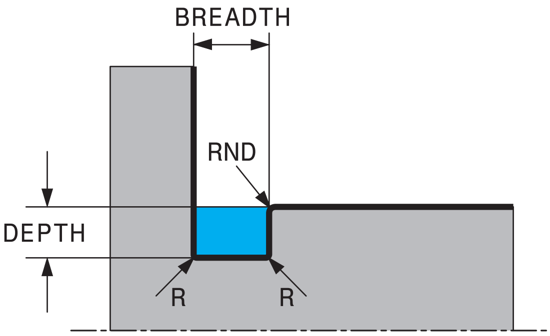
Input parameters in undercut UDC TYPE_K

Parameter	Meaning	Input
R	Corner radius of both inside corners	Required
DEPTH	Undercut depth (parallel to axis)	Required
ROT	Angle relative to longitudinal axis (default: 45°)	Optional
ANG_WIDTH	Angle of undercut opening	Required

Example: Undercut form K with depth = 2, width = 15, opening angle = 30°

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_K R1 DEPTH3 ANG_WIDTH30
14 L  X+60
```

Undercut UDC TYPE_U



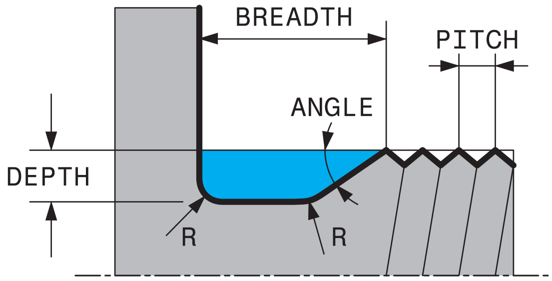
Input parameters in undercut UDC TYPE_U

Parameter	Meaning	Input
R	Corner radius of both inside corners	Required
DEPTH	Undercut depth	Required
BREADTH	Width of undercut	Required
RND / **CHF**	Rounding / chamfer on outside corner	Required

Example: Undercut form U with depth = 3, width = 8

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC TYPE_U R1 DEPTH3 BREADTH8 RND1
14 L  X+60
```

Undercut UDC THREAD



Input parameters in undercut DIN 76 UDC THREAD

Parameter	Meaning	Input
PITCH	Thread pitch	Optional
R	Corner radius of both inside corners	Optional
DEPTH	Undercut depth	Optional
BREADTH	Width of undercut	Optional
ANGLE	Undercut angle	Optional

Example: Thread undercut according to DIN 76 with thread pitch = 2

```
11 L  X+40  Z+0
12 L  Z-30
13 UDC THREAD PITCH2
14 L  X+60
```

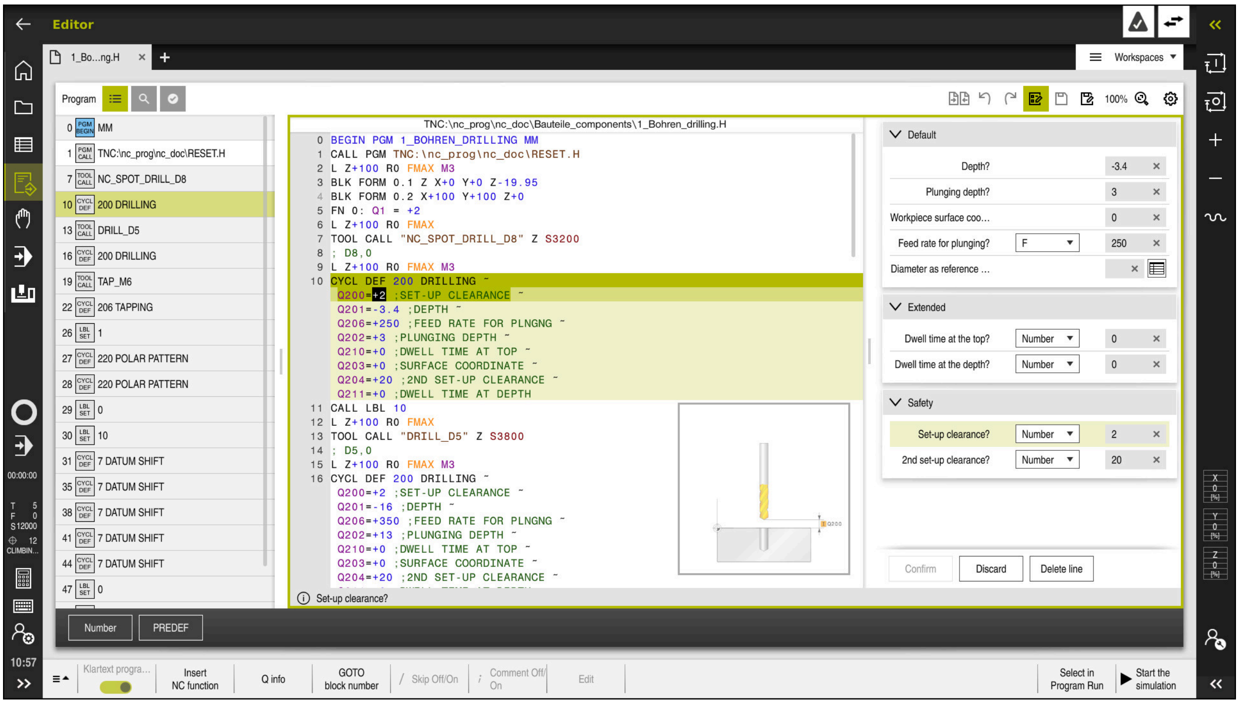
14

Machining Cycles

14.1 Working with machining cycles

14.1.1 Machining cycles

General information



Cycles are stored on the control as subprograms. The cycles can be used to execute different machining operations. This greatly simplifies the task of creating programs. The cycles are also useful for frequently recurring machining operations that comprise several working steps. Most cycles use Q parameters as transfer parameters. The control provides cycles for the following technologies:

- Drilling processes
- Thread machining
- Milling operations such as pockets, studs or even contours
- Cycles for coordinate transformation
- Special cycles
- Turning operations
- Grinding operations

NOTICE

Danger of collision!

Cycles execute extensive operations. Danger of collision!

- Simulate your program before executing it

NOTICE

Danger of collision!

You can program variables as input values in HEIDENHAIN cycles. Using variables outside of the recommended input ranges can lead to collisions.

- Only use the input ranges recommended by HEIDENHAIN
- Pay attention to the HEIDENHAIN documentation
- Check the machining sequence using a simulation

Optional parameters

The comprehensive cycle package is continuously further developed by HEIDENHAIN. Every new software version thus may also introduce new Q parameters for cycles. These new Q parameters are optional parameters, which were not all available in some older software versions. Within a cycle, these parameters are always provided at the end of the cycle definition. You can decide for yourself whether you would like to define optional Q parameters or delete them with the **NO ENT** key. You can also adopt the default value. If you have accidentally deleted an optional Q parameter or if you would like to extend cycles in your existing NC programs, you can add optional Q parameters in cycles where needed. The following steps describe how this is done.

Proceed as follows:

- Call the cycle definition
- Press the right arrow key until the new Q parameters are displayed
- Confirm the displayed default value

  or
- Enter a value
- To load the new Q parameter, exit the menu by selecting the right arrow key once again or by selecting the **END** button
- If you do not wish to load the new Q parameter, press the **NO ENT** key

Compatibility

Most NC programs created with older HEIDENHAIN controls (as of TNC 150 B) can be run with the new software version of the TNC7. Even if new optional parameters have been added to existing cycles, you will generally be able to run your NC programs as usual. This is achieved because the stored default value will be used. The other way around, if you want to run an NC program created with a new software version on an older control, you can delete the respective optional Q parameters from the cycle definition with the **NO ENT** key. In this way you can ensure that the NC program is downward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

14.1.2 Defining cycles

Cycles can be defined in several ways.

Inserting via NC function:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select the desired cycle
- > The control initiates a dialog and prompts you for all required input values.

Inserting via theCYCL DEF key:

CYCL DEF

- Press the **CYCL DEF** key
- > The control opens the **Insert NC function** window.
- Select the desired cycle
- > The control initiates a dialog and prompts you for all required input values.

Navigation in the cycle

Key	Function
▸	Navigation within the cycle: Jump to next parameter
◂	Navigation within the cycle: Jump to previous parameter
▾	Jump to the same parameter in the next cycle
▴	Jump to the same parameter in the previous cycle

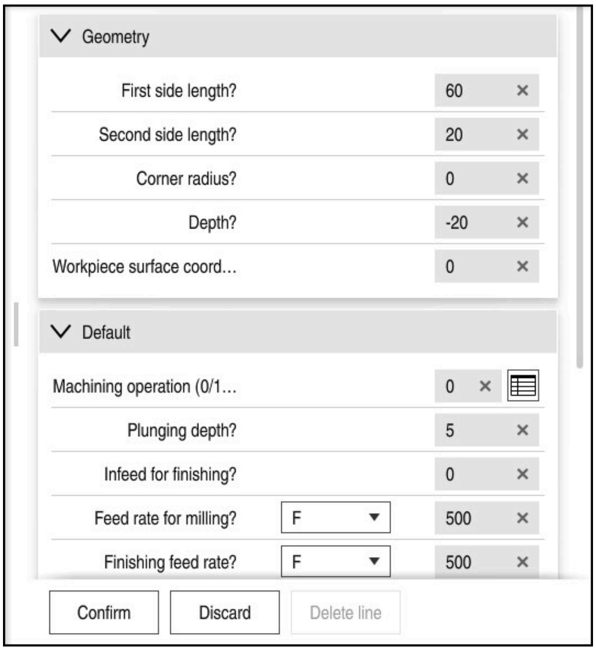
The control provides selection possibilities for the different cycle parameters via the action bar or the form.

If an input option specifying a defined behavior is stored in particular cycle parameters, you can open a selection list with the **GOTO** key or in the form view. For example in cycle **200 DRILLING**, the **Q395 DEPTH REFERENCE** parameter provides the selection possibility:

- 0 | Tool tip
- 1 | Cutting edge corner

Cycle input form

The control offers a **FORM** for various functions and cycles. This **FORM** can be used to enter various syntax elements or also cycle parameters on form-basis.



The control allocates the cycle parameters in the **FORM** to groups based on their functions, e.g. geometry, standard, advanced, safety. The control provides selection possibilities for different cycle parameters via switches, for example. The control displays the currently edited cycle parameter in color.

After you have defined all required cycle parameters, you can confirm your input and conclude the cycle.

Opening the form:

- Open the **Editor** operating mode
- Open the **Program** workspace

- Select **FORM** via the title bar

If an input is invalid, the control displays an information symbol ahead of the syntax element. When you select the information symbol, the control displays information on the error.

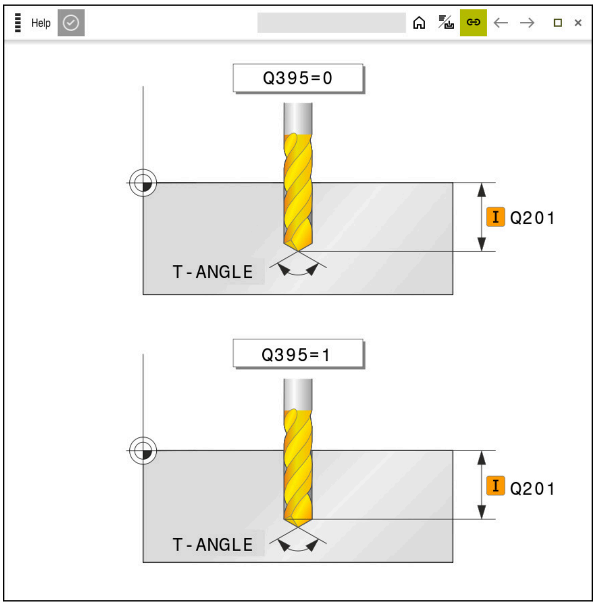
Help graphics

When you are editing a cycle, the control shows a help graphic for the current Q parameters. The size of the help graphic depends on the size of the **Program** workspace area.

The control shows the help graphic at the right edge of the workspace, or at the top or bottom edge. The help graphic is positioned in the half that does not contain the cursor.

When you tap or click on the help graphic, the control maximizes the help graphic.

If the **Help** workspace is active, the control will display the help graphic in this area instead of showing it in the **Program** workspace.



Help workspace with a help graphic for a cycle parameter

14.1.3 Calling cycles

For cycles that remove material, you have to enter not only the cycle definition, but also the cycle call in the NC program. The call always refers to the fixed cycle that was last defined in the NC program.

Requirements

Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (only required for simulation)
- Tool call
- Spindle direction of rotation (miscellaneous function **M3**/**M4**)
- Cycle definition (**CYCL DEF**)

- For some cycles, additional requirements must be observed. They are detailed in the descriptions and overview tables for each cycle.

You can program the cycle call in the following ways.

Option	Further information
CYCL CALL	Page 426
CYCL CALL PAT	Page 426
CYCL CALL POS	Page 427
M89/M99	Page 427

Calling a cycle with CYCL CALL

The **CYCL CALL** function calls the most recently defined fixed cycle once. The starting point of the cycle is the position that was programmed last before the **CYCL CALL** block.

Insert NC function

CYCL CALL

- Select **Insert NC function**
  or
- Press the **CYCL CALL** key
- > The control opens the **Insert NC function** window.
- Select **CYCL CALL M**
- Define **CYCL CALL M** and add an M function , if necessary

Calling a cycle with CYCL CALL PAT

The **CYCL CALL PAT** function calls the most recently defined machining cycle at all positions that you defined in a **PATTERN DEF** pattern definition or in a point table.

Further information: "Pattern definition with PATTERN DEF", Page 365

Further information: "Point tables", Page 346

Insert NC function

CYCL CALL

- Select **Insert NC function**
  or
- Press the **CYCL CALL** key
- > The control opens the **Insert NC function** window.
- Select **CYCL CALL PAT**
- Define **CYCL CALL PAT** and add an M function , if necessary

Calling a cycle with CYCL CALL POS

The **CYCL CALL POS** function calls the most recently defined fixed cycle once. The starting point of the cycle is the position that you defined in the **CYCL CALL POS** block.

Insert NC function

- ▶ Select **Insert NC function**

  or

CYCL CALL

- ▶ Press the **CYCL CALL** key
- > The control opens the **Insert NC function** window.
- ▶ Select **CYCL CALL POS**
- ▶ Define **CYCL CALL POS** and add an M function , if necessary

Using positioning logic, the control moves to the position defined in the **CYCL CALL POS** block:

- If the tool's current position in the tool axis is above the upper edge of the workpiece (**Q203**), the control first moves the tool to the programmed position in the working plane and then to the programmed position in the tool axis
- If the tool's current position in the tool axis is below the upper edge of the workpiece (**Q203**), the control first moves the tool to the clearance height in the tool axis and then to the programmed position in the working plane

> Programming and operating notes
>
> - Three coordinate axes must always be programmed in the **CYCL CALL POS** block. Using the coordinate in the tool axis, you can easily change the starting position. It serves as an additional datum shift.
> - The feed rate most recently defined in the **CYCL CALL POS** block is only used to traverse to the start position programmed in this block.
> - As a rule, the control moves without radius compensation (R0) to the position defined in the **CYCL CALL POS** block.
> - If you use **CYCL CALL POS** to call a cycle in which a start position is defined (e.g. Cycle **212**), then the position defined in the cycle serves as an additional shift of the position defined in the **CYCL CALL POS** block. You should therefore always define the start position in the cycle as 0.

Calling a cycle with M89/M99

The **M99** function, which is active only in the block in which it is programmed (non-modal function), calls the last defined fixed cycle once. You can program **M99** at the end of a positioning block. The control moves to this position and then calls the last defined machining cycle.

If the control is to execute the cycle automatically after every positioning block, program the first cycle call with **M89**.

To cancel the effect of **M89**, proceed as follows:

- ▶ Program **M99** in the positioning block
- > The control moves to the last starting point.

  or
- ▶ Define a new machining cycle with **CYCL DEF**

Defining and calling an NC program as cycle

With **SEL CYCLE**, you can define any NC program as a machining cycle.

Defining an NC program as a cycle:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **SEL CYCLE**
- Select file name, string parameter or file

Calling an NC program as a cycle:

CYCL CALL

- Press the **CYCL CALL** key
- > The control opens the **Insert NC function** window.

  or
- Program **M99**

- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path.
- Please note that **CYCL CALL PAT** and **CYCL CALL POS** use positioning logic before executing the cycle. With respect to the positioning logic, **SEL CYCLE** and Cycle **12 PGM CALL** show the same behavior. In point pattern cycles, the clearance height is calculated based on:
  - the maximum value of all Z positions at the starting point of the pattern
  - all Z positions in the point pattern
- With **CYCL CALL POS**, there will be no pre-positioning in the tool axis direction. This means that you need to manually program any pre-positioning in the file you call.

14.1.4 Machine-specific cycles

Refer to your machine manual for a description of the specific functionality.

Cycles are available for many machines. Your machine manufacturer can implement these cycles into the control, in addition to the HEIDENHAIN cycles. These cycles are available in a separate cycle-number range:

Cycle-number range	Description
300 to **399**	Machine-specific cycles that are to be selected through the **CYCL DEF** key
500 to **599**	Machine-specific touch probe cycles that are to be selected through the **TOUCH PROBE** key

Some machine-specific cycles use transfer parameters that are also part of HEIDENHAIN standard cycles. In order to avoid problems (related to overwriting of transfer parameters that are used more than once), when using DEF-active cycles (cycles that the control runs automatically during cycle definition) and CALL-active cycles (cycles that you need to call to run them) used at the same time,

do the following:

Proceed as follows:

- Program DEF-active cycles before CALL-active cycles

Only program a DEF-active cycle between the definition of a CALL-active cycle and the cycle call if there is no overlapping of transfer parameters in these two cycles.

14.1.5 Available cycle groups

Machining cycles

Cycle group	Further information
Drilling/Thread	
■ Drilling, reaming	Page 433
■ Boring	Page 452
■ Counterboring, centering	
■ Tapping or thread milling	
Pockets/studs/slots	
■ Pocket milling	Page 452
■ Stud milling	
■ Slot milling	
■ Face milling	
Coordinate transformations	
■ Mirroring	Page 958
■ Rotating	
■ Magnifying / Reducing	
SL cycles	
■ SL (Subcontour List) cycles for the machining of contours that possibly consist of several subcontours	Page 452
■ Cylinder surface machining	Page 1178
■ OCM (Optimized Contour Milling) cycles for combining subcontours to form complex contours	Page 392
Point patterns	
■ Bolt hole circle	Page 377
■ Linear hole pattern	
■ Data Matrix code	
Turning cycles	
■ Area clearance cycles, longitudinal and transverse	Page 677
■ Recess turning cycles, radial and axial	
■ Recessing cycles, radial and axial	
■ Thread cutting cycles	
■ Simultaneous turning cycles	
■ Special cycles	

Cycle group	Further information
Special cycles	
■ Dwell time	Page 1126
■ Program call	Page 452
■ Tolerance	Page 898
■ Oriented spindle stop	Page 1148
■ Engraving	
■ Gear cycles	
■ Interpolation turning	
Grinding cycles	
■ Reciprocating stroke	Page 836
■ Dressing	
■ Correction cycles	

Measuring cycles

Cycle group	Further information
Rotation	
■ Probing of plane, edge, two circles, beveled edge ■ Basic rotation ■ Two holes or studs ■ Via rotary axis ■ Via C-axis	Page 1460
Preset/Position	
■ Rectangle, inside or outside ■ Circle, inside or outside ■ Corner, inside or outside ■ Center of bolt circle, slot or ridge ■ Touch probe axis or single axis ■ Four holes	Page 1524
Measuring	
■ Angle ■ Circle, inside or outside ■ Rectangle, inside or outside ■ Slot or ridge ■ Bolt hole circle ■ Plane or coordinate	Page 1605
Special cycles	
■ Measuring or measuring in 3D ■ Probing in 3D ■ Fast probing	Page 1662
Calibrating the touch probe	
■ Calibrating the length ■ Calibration in a ring ■ Calibration on a stud ■ Calibration on a sphere	Page 1679
Measuring kinematics	
■ Saving the kinematics ■ Measure kinematics ■ Preset compensation ■ Kinematics grid	Page 1695
Measuring the tool (TT)	
■ Calibrating the TT ■ Tool length, radius or measuring completely ■ Calibrating the IR-TT ■ Lathe tool measurement	Page 1734

14.2 Technology-independent cycles

14.2.1 Overview

Cycle		Call	Further information
200	**DRILLING** ▪ Basic hole ▪ Input of the dwell time at top and bottom ▪ Depth reference selectable	**CALL**-active	Page 433
201	**REAMING** ▪ Reaming a hole ▪ Input of the dwell time at bottom	**CALL**-active	Page 437
203	**UNIVERSAL DRILLING** ▪ Degression – hole with decreasing infeed ▪ Input of the dwell time at top and bottom ▪ Input of chip breaking behavior ▪ Depth reference selectable	**CALL**-active	Page 439
205	**UNIVERSAL PECKING** ▪ Degression – hole with decreasing infeed ▪ Input of chip breaking behavior ▪ Input of a deepened starting point ▪ Input of an advanced stop distance	**CALL**-active	Page 445

14.2.2 Cycle 200 DRILLING

Application

With this cycle, you can drill basic holes. In this cycle, the depth reference is selectable.

Cycle sequence

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2. The tool drills to the first plunging depth at the programmed feed rate **F**
3. The control retracts the tool at **FMAX** to set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to set-up clearance above the first plunging depth
4. The tool then drills deeper by the plunging depth at the programmed feed rate F.
5. The control repeats this procedure (steps 2 to 4) until the programmed depth is reached (the dwell time from **Q211** is effective with every infeed)
6. Finally, the tool path is retracted from the hole bottom at rapid traverse **FMAX** to setup clearance or to 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

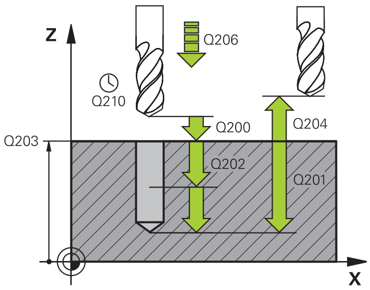
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

If you want to drill without chip breaking, make sure to define, in the **Q202** parameter, a higher value than the depth **Q201** plus the calculated depth based on the point angle. You can enter a much higher value there.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while drilling Input: **0...99999.999** or **FAUTO**, **FU**
	Q202 Plunging depth? Tool infeed per cut. This value has an incremental effect. The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if: ■ the plunging depth is equal to the depth ■ the plunging depth is greater than the depth Input: **0...99999.9999**
	Q210 Dwell time at the top? Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input: **0...3600.0000** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active preset. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**

Help graphic	Parameter
	Q395 Diameter as reference (0/1)? Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T. **0** = Depth referenced to tool tip **1** = Depth referenced to the cylindrical part of the tool Input: **0**, **1**

Example

```
11 CYCL DEF 200 DRILLING ~
    Q200=+2                ;SET-UP CLEARANCE ~
    Q201=-20               ;DEPTH ~
    Q206=+150              ;FEED RATE FOR PLNGNG ~
    Q202=+5                ;PLUNGING DEPTH ~
    Q210=+0                ;DWELL TIME AT TOP ~
    Q203=+0                ;SURFACE COORDINATE ~
    Q204=+50               ;2ND SET-UP CLEARANCE ~
    Q211=+0                ;DWELL TIME AT DEPTH ~
    Q395=+0                ;DEPTH REFERENCE
12 L  X+30  Y+20 FMAX M3
13 CYCL CALL
14 L  X+80  Y+50 FMAX M99
```

14.2.3 Cycle 201 REAMING

Application

With this cycle, you can machine basic fits. In this cycle, you can optionally define a dwell time at the bottom of the hole.

Cycle sequence

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2. The tool reams to the entered depth at the programmed feed rate **F**.
3. If programmed, the tool remains at the hole bottom for the entered dwell time.
4. Then, the control retracts the tool at rapid traverse **FMAX** to setup clearance or to 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

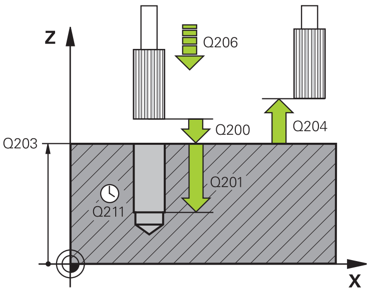
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while reaming Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	Q208 Feed rate for retraction? Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208** = 0, the feed rate for reaming applies. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active preset. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 CYCL DEF 201 REAMING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q201=-20                ;DEPTH ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q211=+0                 ;DWELL TIME AT DEPTH ~
    Q208=+99999             ;RETRACTION FEED RATE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE
12 L  X+30  Y+20 FMAX M3
13 CYCL CALL
```

14.2.4 Cycle 203 UNIVERSAL DRILLING

Application

With this cycle, you can drill holes with decreasing infeed. In this cycle, you can optionally define a dwell time at the bottom of the hole. The cycle may be executed with or without chip breaking.

Cycle sequence

Behavior without chip breaking, without decrement:

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
2. The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3. Then, the control retracts the tool from the hole to **SET-UP CLEARANCE Q200**
4. Now, the control again plunges the tool at rapid traverse into the hole and then again drills an infeed of **PLUNGING DEPTH Q202** at the **FEED RATE FOR PLNGNG Q206**
5. When machining without chip breaking, the control removes the tool from the hole after each infeed at the **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200**. It remains there for the **DWELL TIME AT TOP Q210**
6. This sequence will be repeated until **DEPTH Q201** is reached.
7. When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking, without decrement:

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified **SET-UP CLEARANCE Q200** above the workpiece surface
2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3 Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
4 Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** at the **FEED RATE FOR PLNGNG Q206**
5 The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to **SET-UP CLEARANCE Q200**
6 If programmed, the control will wait for the time specified in **DWELL TIME AT TOP Q210**
7 Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
8 Steps 2 to 7 will be repeated until **DEPTH Q201** is reached
9 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking, with decrement

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SAFETY CLEARANCE Q200** above the workpiece surface
2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
3 Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
4 Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** minus **DECREMENT Q212** at the **FEED RATE FOR PLNGNG Q206**. The increasingly smaller difference between the updated **PLUNGING DEPTH Q202** minus **DECREMENT Q212** must never be smaller than the **MIN. PLUNGING DEPTH Q205** (example: **Q202**=5, **Q212**=1, **Q213**=4, **Q205**= 3: The first plunging depth is 5 mm, the second plunging depth is 5 - 1 = 4 mm, the third plunging depth is 4 - 1 = 3 mm, the fourth plunging depth is also 3 mm)
5 The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to **SET-UP CLEARANCE Q200**
6 If programmed, the control will now wait for the time specified in **DWELL TIME AT TOP Q210**
7 Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
8 Steps 2 to 7 will be repeated until **DEPTH Q201** is reached
9 If programmed, the control will now wait for the time specified in **DWELL TIME AT DEPTH Q211**
10 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to **SET-UP CLEARANCE Q200** or to **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

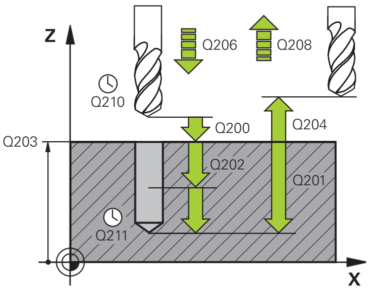
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while drilling Input: **0...99999.999** or **FAUTO**, **FU**
	Q202 Plunging depth? Tool infeed per cut. This value has an incremental effect. The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if: ■ the plunging depth is equal to the depth ■ the plunging depth is greater than the depth Input: **0...99999.9999**
	Q210 Dwell time at the top? Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input: **0...3600.0000** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q212 Decrement? Value by which the control decreases **Q202 Feed depth** after each infeed. This value has an incremental effect. Input: **0...99999.9999**
	Q213 Nr of breaks before retracting? Number of chip breaks after which the control is to withdraw the tool from the hole for chip breaking. For chip breaking, the control retracts the tool each time by the value in **Q256**. Input: **0...99999**

Help graphic	Parameter
	Q205 Minimum plunging depth? If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect. Input: **0...99999.9999**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	Q208 Feed rate for retraction? Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q206**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0...99999.999** or **PREDEF**
	Q395 Diameter as reference (0/1)? Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T. **0** = Depth referenced to tool tip **1** = Depth referenced to the cylindrical part of the tool Input: **0**, **1**

Example

11 CYCL DEF 203 UNIVERSAL DRILLING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q202=+5	;PLUNGING DEPTH ~
Q210=+0	;DWELL TIME AT TOP ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q212=+0	;DECREMENT ~
Q213=+0	;NR OF BREAKS ~
Q205=+0	;MIN. PLUNGING DEPTH ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q208=+99999	;RETRACTION FEED RATE ~
Q256=+0.2	;DIST FOR CHIP BRKNG ~
Q395=+0	;DEPTH REFERENCE
12 L X+30 Y+20 FMAX M3	
13 CYCL CALL	

14.2.5 Cycle 205 UNIVERSAL PECKING

Application

With this cycle, you can drill holes with decreasing infeed. The cycle may be executed with or without chip breaking. When the plunging depth is reached the cycle performs chip removal. If there is already a pilot hole then you can enter a deepened starting point. In this cycle, you can optionally define a dwell time at the bottom of the hole. This dwell time is used for chip breaking at the bottom of the hole.

Further information: "Chip removal and chip breaking", Page 450

Cycle sequence

1 The control positions the tool in the tool axis at **FMAX** to the entered **Safety clearance Q200** above the **SURFACE COORDINATE Q203**.
2 If you program a deepened starting point in **Q379**, the control moves at the positioning feed rate **Q253 F PRE-POSITIONING** to the set-up clearance above the deepened starting point.
3 The tool drills at the programmed **Q206 FEED RATE FOR PLNGNG** to the plunging depth.
4 If you have programmed chip breaking, the control retracts the tool by the retraction value **Q256**.
5 Upon reaching the plunging depth, the control retracts the tool in the tool axis at the retraction feed rate **Q208** to the set-up clearance. The set-up clearance is above the **SURFACE COORDINATE Q203**.
6 The tool then moves at **Q373 FEED AFTER REMOVAL** to the entered advanced stop distance above the plunging depth last reached.
7 The tool drills at the feed in **Q206** to the next plunging depth. If a decrement Q212 is defined, the plunging depth is decreased after each infeed by the decrement.
8 The control repeats this procedure (steps 2 to 7) until the total drilling depth is reached.
9 If you entered a dwell time, the tool remains at the hole bottom for chip breaking. The control then retracts the tool at the retraction feed rate to the set-up clearance or the 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**.

> After chip removal, the depth of the next chip breaking is referenced to the last plunging depth.
>
> **Example:**
>
> - **Q202 PLUNGING DEPTH** = 10 mm
> - **Q257 DEPTH FOR CHIP BRKNG** = 4 mm
>
> The control performs chip breaking at 4 mm and 8 mm. Chip removal is performed at 10 mm. Chip breaking is next performed at 14 mm and 18 mm, etc.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

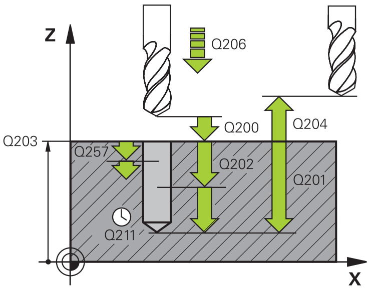
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

This cycle is not suitable for overlong drills. For overlong drills, use Cycle **241 SINGLE-LIP D.H.DRLNG**.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- If you enter advance stop distances **Q258** not equal to **Q259**, the control will change the advance stop distances between the first and last plunging depths at the same rate.
- If you use **Q379** to enter a deepened starting point, the control will change the starting point of the infeed movement. Retraction movements are not changed by the control; they are always calculated with respect to the coordinate of the workpiece surface.
- If **Q257 DEPTH FOR CHIP BRKNG** is greater than **Q202 PLUNGING DEPTH**, the operation is executed without chip breaking.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of hole (depends on parameter **Q395 DEPTH REFERENCE**). This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while drilling Input: **0...99999.999** or **FAUTO**, **FU**
	Q202 Plunging depth? Tool infeed per cut. This value has an incremental effect. The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if: ■ the plunging depth is equal to the depth ■ the plunging depth is greater than the depth Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q212 Decrement? Value by which the control decreases the plunging depth **Q202**. This value has an incremental effect. Input: **0...99999.9999**
	Q205 Minimum plunging depth? If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q258 Upper advanced stop distance? Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after first chip removal. This value has an incremental effect. Input: **0...99999.9999**
	Q259 Lower advanced stop distance? Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after last chip removal. This value has an incremental effect. Input: **0...99999.9999**
	Q257 Infeed depth for chip breaking? Incremental depth at which the control performs chip breaking. This procedure is repeated until **DEPTH Q201** is reached. If **Q257** equals 0, the control will not perform chip breaking. This value has an incremental effect. Input: **0...99999.9999**
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0...99999.999** or **PREDEF**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	Q379 Deepened starting point? If there is already a pilot hole then you can define a deepened starting point here. It is incrementally referenced to **Q203 SURFACE COORDINATE**. The control moves at **Q253 F PRE-POSITIONING** to above the deepened starting point by the value **Q200 SET-UP CLEARANCE**. This value has an incremental effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Defines the tool traversing speed when positioning from **Q200 SET-UP CLEARANCE** to **Q379 STARTING POINT** (not equal to 0). Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q208 Feed rate for retraction? Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q206**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q395 Diameter as reference (0/1)? Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T. **0** = Depth referenced to tool tip **1** = Depth referenced to the cylindrical part of the tool Input: **0**, **1**
	Q373 Post-chip-removal approach feed? Traversing speed of the tool when approaching the advanced stop distance after chip removal. **0**: Move at **FMAX** **>0**: Feed in mm/min Input: **0...99999** or **FAUTO**, **FMAX**, **FU**, **FZ**

Example

```
11 CYCL DEF 205 UNIVERSAL PECKING ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q201=-20            ;DEPTH ~
    Q206=+150           ;FEED RATE FOR PLNGNG ~
    Q202=+5             ;PLUNGING DEPTH ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q212=+0             ;DECREMENT ~
    Q205=+0             ;MIN. PLUNGING DEPTH ~
    Q258=+0.2           ;UPPER ADV STOP DIST ~
    Q259=+0.2           ;LOWER ADV STOP DIST ~
    Q257=+0             ;DEPTH FOR CHIP BRKNG ~
    Q256=+0.2           ;DIST FOR CHIP BRKNG ~
    Q211=+0             ;DWELL TIME AT DEPTH ~
    Q379=+0             ;STARTING POINT ~
    Q253=+750           ;F PRE-POSITIONING ~
    Q208=+99999         ;RETRACTION FEED RATE ~
    Q395=+0             ;DEPTH REFERENCE ~
    Q373=+0             ;FEED AFTER REMOVAL
```

Chip removal and chip breaking

Chip removal

Chip removal depends on cycle parameter **Q202 PLUNGING DEPTH**.

When the value entered in cycle parameter **Q202** is reached, the control performs chip removal. This means that the control always moves the tool to the retraction height, irrespective of the deepened starting point **Q379**. This height is calculated from **Q200 SET-UP CLEARANCE** + **Q203 SURFACE COORDINATE**

Example:

0 BEGIN PGM 205 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 203 Z S4500	; Tool call (tool radius 3)
4 L Z+250 R0 FMAX	; Retract the tool
5 CYCL DEF 205 UNIVERSAL PECKING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-20 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q212=+0 ;DECREMENT ~	
Q205=+0 ;MIN. PLUNGING DEPTH ~	
Q258=+0.2 ;UPPER ADV STOP DIST ~	
Q259=+0.2 ;LOWER ADV STOP DIST ~	
Q257=+0 ;DEPTH FOR CHIP BRKNG ~	
Q256=+0.2 ;DIST FOR CHIP BRKNG ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q379=+10 ;STARTING POINT ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q208=+3000 ;RETRACTION FEED RATE ~	
Q395=+0 ;DEPTH REFERENCE ~	
Q373=+0 ;FEED AFTER REMOVAL	
6 L X+30 Y+30 R0 FMAX M3	; Approach drilling position, spindle ON
7 CYCL CALL	; Cycle call
8 L Z+250 R0 FMAX	; Retract the tool, end program
9 M30	
10 END PGM 205 MM	

Chip breaking

Chip breaking depends on cycle parameter **Q257 DEPTH FOR CHIP BRKNG**.

When the value entered in cycle parameter **Q257** is reached, the control performs chip breaking. This means that the control retracts the tool by the value defined in **Q256 DIST FOR CHIP BRKNG**. Chip removal starts once the tool reaches the **PLUNGING DEPTH**. The entire process is repeated until **DEPTH Q201** is reached.

Example:

0 BEGIN PGM 205 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 203 Z S4500	; Tool call (tool radius 3)
4 L Z+250 R0 FMAX	; Retract the tool
5 CYCL DEF 205 UNIVERSAL PECKING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-20 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+10 ;PLUNGING DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q212=+0 ;DECREMENT ~	
Q205=+0 ;MIN. PLUNGING DEPTH ~	
Q258=+0.2 ;UPPER ADV STOP DIST ~	
Q259=+0.2 ;LOWER ADV STOP DIST ~	
Q257=+3 ;DEPTH FOR CHIP BRKNG ~	
Q256=+0.5 ;DIST FOR CHIP BRKNG ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q379=+0 ;STARTING POINT ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q208=+3000 ;RETRACTION FEED RATE ~	
Q395=+0 ;DEPTH REFERENCE ~	
Q373=+0 ;FEED AFTER REMOVAL	
6 L X+30 Y+30 R0 FMAX M3	; Approach drilling position, spindle ON
7 CYCL CALL	; Cycle call
8 L Z+250 R0 FMAX	; Retract the tool, end program
9 M30	
10 END PGM 205 MM	

14.3 Cycles for milling

14.3.1 Overview

Cycle		Call	Further information
202	**BORING** ■ Boring a hole ■ Input of the retraction feed rate ■ Input of the dwell time at bottom ■ Input of the retracting movement	CALL-active	Page 455
204	**BACK BORING** ■ Machining a counterbore on the underside of the workpiece ■ Input of the dwell time ■ Input of the retracting movement	CALL-active	Page 458
208	**BORE MILLING** ■ Milling of a hole ■ Input of a pre-drill diameter ■ Climb or up-cut milling selectable	CALL-active	Page 463
241	**SINGLE-LIP D.H.DRLNG** ■ Drilling with single-lip deep hole drill ■ Deepened starting point ■ Direction of rotation and rotational speed for moving into and retracting from the hole ■ Input of the dwell depth	CALL-active	Page 466
240	**CENTERING** ■ Drilling a center hole ■ Input of the centering diameter or depth ■ Input of the dwell time at bottom	CALL-active	Page 476
206	**TAPPING** ■ With a floating tap holder ■ Input of the dwell time at bottom	CALL-active	Page 478
207	**RIGID TAPPING** ■ Without a floating tap holder ■ Input of the dwell time at bottom	CALL-active	Page 481
209	**TAPPING W/ CHIP BRKG** ■ Without a floating tap holder ■ Input of chip breaking behavior	CALL-active	Page 484
262	**THREAD MILLING** ■ Milling a thread into pre-drilled material	CALL-active	Page 490
263	**THREAD MLLNG/CNTSNKG** ■ Milling a thread into pre-drilled material ■ Machining a countersunk chamfer	CALL-active	Page 494
264	**THREAD DRILLNG/MLLNG**	CALL-active	Page 499

Cycle		Call	Further information
	■ Drilling into solid material ■ Milling a thread		
265	**HEL. THREAD DRLG/MLG** ■ Milling a thread into solid material	**CALL-** active	Page 504
267	**OUTSIDE THREAD MLLNG** ■ Milling an external thread ■ Machining a countersunk chamfer	**CALL-** active	Page 508
251	**RECTANGULAR POCKET** ■ Roughing and finishing cycle ■ Plunging strategy: helical, reciprocating, or vertical	**CALL-** active	Page 512
252	**CIRCULAR POCKET** ■ Roughing and finishing cycle ■ Plunging strategy: helical or vertical	**CALL-** active	Page 518
253	**SLOT MILLING** ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	**CALL-** active	Page 524
254	**CIRCULAR SLOT** ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	**CALL-** active	Page 529
256	**RECTANGULAR STUD** ■ Roughing and finishing cycle ■ Approach position: selectable	**CALL-** active	Page 536
257	**CIRCULAR STUD** ■ Roughing and finishing cycle ■ Input of the start angle ■ Helical infeed starting from the workpiece blank diameter	**CALL-** active	Page 542
258	**POLYGON STUD** ■ Roughing and finishing cycle ■ Helical infeed starting from the workpiece blank diameter	**CALL-** active	Page 547
233	**FACE MILLING** ■ Roughing and finishing cycle ■ Roughing strategy and direction: selectable ■ Input of side walls	**CALL-** active	Page 552
20	**CONTOUR DATA** ■ Input of machining information	**DEF-** active	Page 565
21	**PILOT DRILLING** ■ Machining a hole for non-center cutting tools	**CALL-** active	Page 567
22	**ROUGH-OUT** ■ Roughing or fine roughing of the contour ■ Takes infeed points of the rough-out tool into account	**CALL-** active	Page 569

Cycle		Call	Further information
23	**FLOOR FINISHING** ■ Finishing with finishing allowance for the floor from Cycle **20**	**CALL**-active	Page 572
24	**SIDE FINISHING** ■ Finishing with side finishing allowance from Cycle **20**	**CALL**-active	Page 578
270	**CONTOUR TRAIN DATA** ■ Input of contour data for Cycle **25** or **276**	**DEF**-active	Page 578
25	**CONTOUR TRAIN** ■ Machining of open and closed contours ■ Monitoring for undercuts and contour damage	**CALL**-active	Page 580
275	**TROCHOIDAL SLOT** ■ Machining of open and closed contours using trochoidal milling.	**CALL**-active	Page 584
276	**THREE-D CONT. TRAIN** ■ Machining of open and closed contours ■ Detection of residual material ■ 3D contours—additional processing of coordinates from the tool axis	**CALL**-active	Page 590
271	**OCM CONTOUR DATA** (option 167) ■ Definition of the machining information for the contour or subprograms ■ Input of a bounding frame or block	**DEF**-active	Page 596
272	**OCM ROUGHING** (option 167) ■ Technology data for roughing contours ■ Use of the OCM cutting data calculator ■ Plunging behavior: vertical, helical, or reciprocating ■ Plunging strategy: selectable	**CALL**-active	Page 598
273	**OCM FINISHING FLOOR** (option 167) ■ Finishing with finishing allowance for the floor from Cycle **271** ■ Machining strategy with constant tool angle or with path calculated as equidistant (equal distances)	**CALL**-active	Page 613
274	**OCM FINISHING SIDE** (option 167) ■ Finishing with side finishing allowance from Cycle **271**	**CALL**-active	Page 616
277	**OCM CHAMFERING** (option 167) ■ Deburr the edges ■ Consider adjacent contours and walls	**CALL**-active	Page 618
291	**COUPLG.TURNG.INTERP.** (option 96) ■ Coupling of the tool spindle with the positions of the linear axes ■ Or, rescind the spindle coupling	**CALL**-active	Page 621

Cycle		Call	Further information
292	**CONTOUR.TURNG.INTRP.** (option 96) ■ Coupling of the tool spindle with the positions of the linear axes ■ Create certain rotationally symmetric contours in the active working plane ■ Possible with tilted working plane	**CALL**-active	Page 628
225	**ENGRAVING** ■ Engrave texts on a plane surface ■ Arranged in a straight line or along a circular arc	**CALL**-active	Page 638
232	**FACE MILLING** ■ Face mill a level surface in multiple infeeds ■ Selection of the milling plan	**CALL**-active	Page 645
18	**THREAD CUTTING** ■ With controlled spindle ■ Spindle stops at the bottom of the hole	**CALL**-active	Page 651

14.3.2 Cycle 202 BORING

Application

> Refer to your machine manual.
> Machine and control must be specially prepared by the machine manufacturer for use of this cycle.
> This cycle is effective only for machines with servo-controlled spindle.

With this cycle, you can bore holes. In this cycle, you can optionally define a dwell time at the bottom of the hole.

Cycle sequence

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the safety clearance **Q200** above the workpiece **Q203 SURFACE COORDINATE**
2. The tool drills to the programmed depth at the feed rate for plunging **Q201**
3. If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
4. The control then carries out an oriented spindle stop to the position that is defined in the **Q336** parameter
5. If **Q214 DISENGAGING DIRECTN** is defined, the control retracts in the programmed direction by the value in **CLEARANCE TO SIDE Q357**
6. Then the control moves the tool at the retraction feed rate **Q208** to the set-up clearance **Q200**
7. The tool is again centered in the hole
8. The control restores the spindle status as it was at the cycle start.
9. If programmed, the control moves the tool at **FMAX** to 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**. If **Q214**=0 the tool tip remains on the wall of the hole

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

- Check the position of the tool tip when programming an oriented spindle stop with reference to the angle entered in **Q336** (e.g. in the **MDI** application in the **Manual** operating mode). In this case, no transformations should be active.
- Select the angle so that the tool tip is parallel to the disengaging direction
- Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

NOTICE

Danger of collision!

If you have activated **M136**, the tool will not move to the programmed set-up clearance once the machining operation is finished. The spindle rotation will stop at the bottom of the hole which, in turn, also stops the feed motion. There is a danger of collision as the tool will not be retracted!

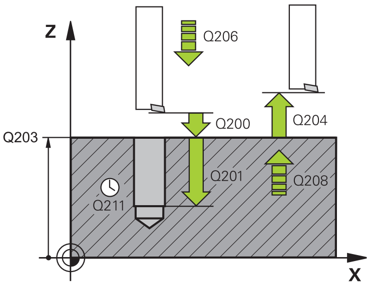
- Use **M137** to deactivate **M136** before the cycle start

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- If **Q214 DISENGAGING DIRECTN** is not 0, **Q357 CLEARANCE TO SIDE** is in effect.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while boring Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	Q208 Feed rate for retraction? Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208**=0, the feed rate for plunging applies. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q214 Disengaging directn (0/1/2/3/4)? Specify the direction in which the control retracts the tool at the hole bottom (after carrying out an oriented spindle stop) **0**: Do not retract tool **1**: Retract tool in negative main axis direction **2**: Retract tool in negative secondary axis direction **3**: Retract tool in positive main axis direction **4**: Retract tool in positive secondary axis direction Input: **0**, **1**, **2**, **3**, **4**
	Q336 Angle for spindle orientation? Angle to which the control positions the tool before retracting it. The value has an absolute effect. Input: **0...360**

Help graphic	Parameter
	Q357 Safety clearance to the side? Distance between tool tooth and the wall. This value has an incremental effect. Only in effect if **Q214 DISENGAGING DIRECTN** is not 0. Input: **0...99999.9999**

Example

11 L Z+100 R0 FMAX	
12 CYCL DEF 202 BORING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q208=+99999	;RETRACTION FEED RATE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q214=+0	;DISENGAGING DIRECTN ~
Q336=+0	;ANGLE OF SPINDLE ~
Q357+0.2	;CLEARANCE TO SIDE
13 L X+30 Y+20 FMAX M3	
14 CYCL CALL	
15 L X+80 Y+50 FMAX M99	

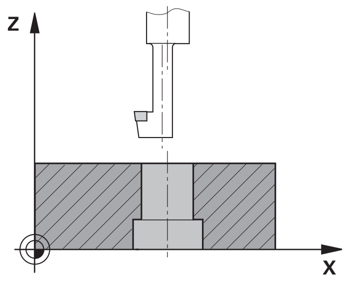
14.3.3 Cycle 204 BACK BORING

Application

Refer to your machine manual.
Machine and control must be specially prepared by the machine manufacturer for use of this cycle.
This cycle is effective only for machines with servo-controlled spindle.

Special boring bars for upward cutting are required for this cycle.

This cycle allows counterbores to be machined from the underside of the workpiece.



Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface
2 The control then orients the spindle to the 0° position with an oriented spindle stop, and displaces the tool by the off-center distance.
3 The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the cutting edge has reached the programmed set-up clearance beneath the lower workpiece edge
4 The control then centers the tool again in the bore hole, switches on the spindle and, if applicable, the coolant and moves the tool at the feed rate for counterboring to the depth programmed for the counterbore
5 If programmed, the tool remains at the counterbore bottom. The tool will then be retracted from the hole again. The control carries out another oriented spindle stop and the tool is once again displaced by the off-center distance
6 Finally the tool moves at **FMAX** to set-up clearance.
7 The tool is again centered in the hole
8 The control restores the spindle status as it was at the cycle start.
9 If necessary, the control moves the tool to 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Notes

NOTICE

Danger of collision!

There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

- ▶ Check the position of the tool tip when programming an oriented spindle stop with reference to the angle entered in **Q336** (e.g. in the **MDI** application in the **Manual** operating mode). In this case, no transformations should be active.
- ▶ Select the angle so that the tool tip is parallel to the disengaging direction
- ▶ Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

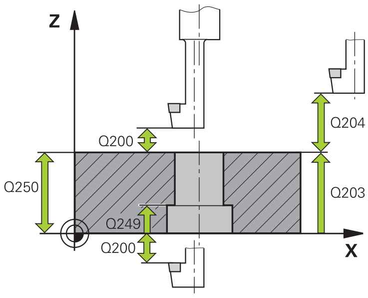
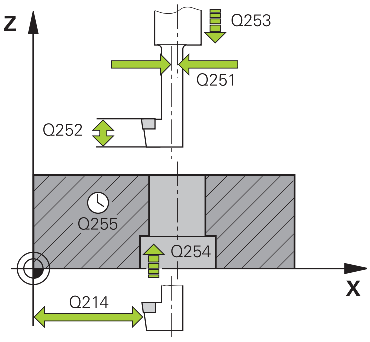
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- When calculating the starting point for boring, the control considers the cutting edge length of the boring bar and the thickness of the material.
- If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF COUNTERBORE Q249**, the control will display an error message.

Enter the tool length measured up to the lower edge of the boring bar, not the cutting edge.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the cycle parameter depth determines the working direction. Note: If you enter a positive sign, the tool bores in the direction of the positive spindle axis.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q249 Depth of counterbore? Distance between underside of workpiece and the top of hole. A positive sign means the hole will be bored in the positive spindle axis direction. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q250 Material thickness? Height of the workpiece. Enter an incremental value. Input: **0.0001...99999.9999**
	**Q251 Tool edge off-center distance?** Off-center distance of the boring bar. Refer to the tool data sheet. This value has an incremental effect. Input: **0.0001...99999.9999**
	Q252 Tool edge height? Distance between underside of boring bar and main cutting tooth. Refer to the tool data sheet. This value has an incremental effect.
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q254 Feed rate for counterboring? Traversing speed of the tool in mm/min during counterboring Input: **0...99999.999** or **FAUTO**, **FU**
	Q255 Dwell time in secs.? Dwell time in seconds at the bottom of the bore hole Input: **0...99999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q214 Disengaging directn (0/1/2/3/4)? Specify the direction in which the control offsets the tool by the off-center distance (after orienting the spindle). Inputting 0 is not permitted **1**: Retract tool in negative main axis direction **2**: Retract tool in negative secondary axis direction **3**: Retract tool in positive main axis direction **4**: Retract tool in positive secondary axis direction Input: **1**, **2**, **3**, **4**
	Q336 Angle for spindle orientation? Angle at which the control positions the tool before it is plunged into or retracted from the bore hole The value has an absolute effect. Input: **0...360**

Example

```
11 CYCL DEF 204 BACK BORING ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q249=+5             ;DEPTH OF COUNTERBORE ~
    Q250=+20            ;MATERIAL THICKNESS ~
    Q251=+3.5           ;OFF-CENTER DISTANCE ~
    Q252=+15            ;TOOL EDGE HEIGHT ~
    Q253=+750           ;F PRE-POSITIONING ~
    Q254=+200           ;F COUNTERBORING ~
    Q255=+0             ;DWELL TIME ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q214=+0             ;DISENGAGING DIRECTN ~
    Q336=+0             ;ANGLE OF SPINDLE
12 CYCL CALL
```

14.3.4 Cycle 208 BORE MILLING

Application

With this cycle, you can mill holes. In this cycle, you can define an optional, pre-drilled diameter. You can also program tolerances for the nominal diameter.

Cycle sequence

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance **Q200** above the workpiece surface
2. The control moves on a semicircle for the first helical path while considering the path overlap **Q370**. The semicircle begins at the center of the hole.
3. The tool mills in a helix to the entered drilling depth at the programmed feed rate **F**.
4. When the drilling depth is reached, the control once again traverses a full circle to remove the material remaining after the initial plunge.
5. The control then centers the tool in the hole again and retracts it to set-up clearance **Q200**.
6. This procedure is repeated until the nominal diameter is reached (the control calculates the stepover by itself)
7. Finally, the tool is retracted to the set-up clearance or to the 2nd set-up clearance **Q204** at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

> If you program **Q370=0** for the path overlap, the control uses the greatest path overlap possible for the first helical path. The control does this to prevent the tool from contacting the workpiece surface. All other paths are distributed uniformly.

Tolerances

The control allows you to store tolerances in the parameter **Q335 NOMINAL DIAMETER**.
You can define the following tolerances:

Tolerance	Example	Manufacturing dimension
Deviations	10+0.01-0.015	9.9975
DIN EN ISO 286-2	10H7	10.0075
ISO 2768-1	10m	10.0000

Proceed as follows:

- Start cycle definition
- Define cycle parameters
- Select the **TEXT** selection possibility in the action bar
- Enter a nominal dimension including tolerance

> - Machining is performed at mid-tolerance.
> - If you program an incorrect tolerance, the control interrupts machining with an error message.
> - Pay attention to capitalization when entering tolerances.

Notes

NOTICE
Danger of collision! If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! ▶ Enter depth as negative ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

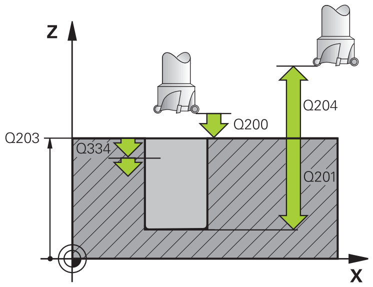
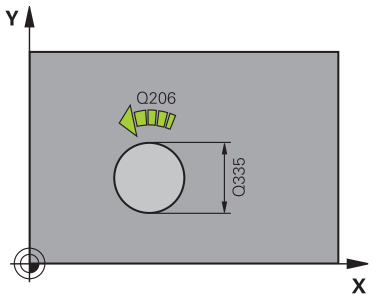
NOTICE
Caution: Danger to the workpiece and tool! If the selected infeed is too large, there is a danger of tool breakage and damage to the workpiece. ▶ Specify the maximum possible plunge angle and the corner radius **DR2** in the **ANGLE** column of the **TOOL.T** tool table. > The control automatically calculates the max. permissible infeed and changes your entered value accordingly, if necessary.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you have entered the bore hole diameter to be the same as the tool diameter, the control will bore directly to the entered depth without any helical interpolation.
- An active mirror function **does not** influence the type of milling defined in the cycle.
- When calculating the overlap factor, the control takes the corner radius **DR2** of the current tool into account.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues an error message.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
 	**Q200 Set-up clearance?** Distance between lower edge of tool and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min during helical drilling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
 	**Q334 Feed per revolution of helix** Depth of the tool plunge with each helix (=360°). This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q335 Nominal diameter? Hole diameter. If you entered the nominal diameter to be the same as the tool diameter, the control will bore directly to the entered depth without any helical interpolation. The value has an absolute effect. You can program a tolerance if needed. **Further information:** "Tolerances", Page 463 Input: **0...99999.9999**
	Q342 Roughing diameter? Enter the dimension of the pre-drilled diameter. The value has an absolute effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (if you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q370 Path overlap factor? The control uses the path overlap factor to determine the stepover factor k. **0**: The control uses the greatest path overlap possible for the first helical path. The control does this to prevent the tool from contacting the workpiece surface. All other paths are distributed uniformly. **>0**: The control multiplies the factor by the active tool radius. The result is the stepover factor k. Input: **0.1...1999** or **PREDEF**

Example

```
11 CYCL DEF 208 BORE MILLING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q201=-20                ;DEPTH ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q334=+0.25              ;PLUNGING DEPTH ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q335=+5                 ;NOMINAL DIAMETER ~
    Q342=+0                 ;ROUGHING DIAMETER ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q370=+0                 ;TOOL PATH OVERLAP
12 CYCL CALL
```

14.3.5 Cycle 241 SINGLE-LIP D.H.DRLNG

Application

Cycle **241 SINGLE-LIP D.H.DRLNG** enables you to machine holes using a single-lip deep hole drill. It is possible to enter a deepened starting point. You can define the direction of rotation and the rotational speed for moving into and retracting from the hole.

Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **Safety clearance Q200** above the workpiece **SURFACE COORDINATE Q203**
2 Depending on the "Position behavior when working with Q379", Page 472, the control will either switch on the spindle with the programmed speed at the **Safety clearance Q200** or at a certain distance above the coordinate surface
3 The control executes the approach motion depending on the direction of rotation defined in the cycle with a spindle that rotates clockwise, counterclockwise, or is stationary
4 The tool drills to the hole depth at the feed rate **F**, or to the maximum plunging depth if a smaller infeed value has been entered. The plunging depth is decreased after each infeed by the decrement. If you have entered a dwell depth, the control reduces the feed rate by the feed rate factor after the dwell depth has been reached
5 If programmed, the tool remains at the hole bottom for chip breaking.
6 The control repeats this procedure (steps 4 to 5) until the total hole depth is reached
7 After the control has reached this position, it will automatically switch off the coolant and set the speed to the value defined in **Q427 ROT.SPEED INFEED/OUT**
8 The control positions the tool to the retract position at the retraction feed rate. To find out the retract position value in your particular case, please refer to: Page 472
9 If programmed, the tool moves to 2nd set-up clearance at **FMAX**

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

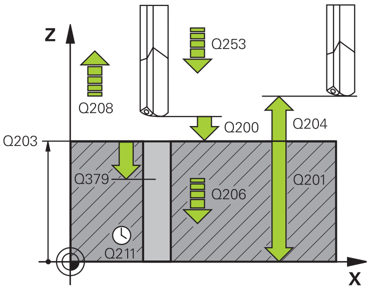
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and **Q203 SURFACE COORDINATE**. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth? Distance between **Q203 SURFACE COORDINATE** and bottom of hole. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while drilling Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active preset. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q379 Deepened starting point? If there is already a pilot hole then you can define a deepened starting point here. It is incrementally referenced to **Q203 SURFACE COORDINATE**. The control moves at **Q253 F PRE-POSITIONING** to above the deepened starting point by the value **Q200 SET-UP CLEARANCE**. This value has an incremental effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q208 Feed rate for retraction? Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208**=0, the control retracts the tool at **Q206 FEED RATE FOR PLNGNG**. Input: **0...99999.999** or **FMAX**, **FAUTO**, **PREDEF**
	Q426 Rot. dir. of entry/exit (3/4/5)? Rotational speed at which the tool is to rotate when moving into and retracting from the hole. **3**: Spindle rotation with M3 **4**: Spindle rotation with M4 **5**: Movement with stationary spindle Input: **3**, **4**, **5**
	Q427 Spindle speed of entry/exit? Rotational speed at which the tool is to rotate when moving into and retracting from the hole. Input: **0...99999**
	Q428 Spindle speed for drilling? Desired speed for drilling. Input: **0...99999**
	Q429 M function for coolant on? **>=0**: Miscellaneous function M for switching on the coolant. The control switches the coolant on when the tool has reached the set-up clearance **Q200** above the starting point **Q379**. **"..."**: Path of a user macro that is to be executed instead of an M function. All instructions in the user macro are executed automatically. **Further information:** "User macro", Page 471 Input: **0...999**
	Q430 M function for coolant off? **>=0**: Miscellaneous function M for switching off the coolant. The control switches the coolant off if the tool is at the **DEPTH Q201**. **"..."**: Path of a user macro that is to be executed instead of an M function. All instructions in the user macro are executed automatically. **Further information:** "User macro", Page 471 Input: **0...999**

Help graphic	Parameter
	Q435 Dwell depth? Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (default setting). Application: During machining of through-holes some tools require a short dwell time before leaving the bottom of the hole in order to transport the chips to the top. Define a value smaller than **Q201 DEPTH**. This value has an incremental effect. Input: **0...99999.9999**
	Q401 Feed rate factor in %? Factor by which the control reduces the feed rate after reaching **Q435 DWELL DEPTH**. Input: **0.0001...100**
	Q202 Maximum plunging depth? Infeed per cut. The **DEPTH Q201** does not have to be a multiple of **Q202**. This value has an incremental effect. Input: **0...99999.9999**
	Q212 Decrement? Value by which the control decreases **Q202 Feed depth** after each infeed. This value has an incremental effect. Input: **0...99999.9999**
	Q205 Minimum plunging depth? If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect. Input: **0...99999.9999**

Example

```
11 CYCL DEF 241 SINGLE-LIP D.H.DRLNG ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q201=-20                ;DEPTH ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q211=+0                 ;DWELL TIME AT DEPTH ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q379=+0                 ;STARTING POINT ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q208=+1000              ;RETRACTION FEED RATE ~
    Q426=+5                 ;DIR. OF SPINDLE ROT. ~
    Q427=+50                ;ROT.SPEED INFEED/OUT ~
    Q428=+500               ;ROT. SPEED DRILLING ~
    Q429=+8                 ;COOLANT ON ~
    Q430=+9                 ;COOLANT OFF ~
    Q435=+0                 ;DWELL DEPTH ~
    Q401=+100               ;FEED RATE FACTOR ~
    Q202=+99999             ;MAX. PLUNGING DEPTH ~
    Q212=+0                 ;DECREMENT ~
    Q205=+0                 ;MIN. PLUNGING DEPTH
12 CYCL CALL
```

User macro

The user macro is another NC program.

A user macro contains a sequence of multiple instructions. With a macro, you can define multiple NC functions that the control executes. As a user, you create macros as an NC program.

Macros work in the same manner as NC programs that are called with the **PGM CALL** function, for example. You define a macro as an NC program with the file type *.h.

- HEIDENHAIN recommends using QL parameters in the macro. QL parameters have only a local effect for an NC program. If you use other types of variables in the macro, then changes may also have an effect on the calling NC program. In order to explicitly cause changes in the calling NC program, use Q or QS parameters with the numbers 1200 to 1399.
- Within the macro, you can read the value of the cycle parameters.

  Further information: "Variables: Q, QL, QR and QS parameters", Page 1268

Example of a user macro for coolant

0 BEGIN PGM KM MM	
1 FN 18: SYSREAD QL100 = ID20 NR8	; Read the coolant level
2 FN 9: IF +QL100 EQU +1 GOTO LBL "Start"	; Query the coolant level; if coolant is active, jump to the **Start** LBL
3 M8	; Switch coolant on
7 CYCL DEF 9.0 DWELL TIME	
8 CYCL DEF 9.1 V.ZEIT3	
9 LBL "Start"	
10 END PGM RET MM	

Position behavior when working with Q379

Especially when working with very long drills, e.g. single-lip deep hole drills or overlong twist drills, there are several things to remember. The position at which the spindle is switched on is very important. If the tool is not guided properly, overlong drills might break.

It is therefore advisable to use the **STARTING POINT Q379** parameter. This parameter can be used to influence the position at which the control turns on the spindle.

Start of drilling

The **STARTING POINT Q379** parameter takes both **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter into account. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The control switches on the spindle at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

STARTING POINT Q379>0

The starting point is at a certain value above the deepened starting point **Q379**. This value can be calculated as follows: 0.2 x **Q379**; if the result of this calculation is larger than **Q200**, the value is always **Q200**.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCE Q200** =2
- **STARTING POINT Q379** =2

The starting point of drilling is calculated as follows: 0.2 x **Q379**=0.2*2=0.4; the starting point is 0.4 mm or inches above the deepened starting point. So if the deepened starting point is at –2, the control starts the drilling process at –1.6.

The following table shows various examples for calculating the start of drilling:

Start of drilling at deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.2 * Q379	Start of drilling
2	2	0	2	0.2*2=0.4	-1.6
2	5	0	2	0.2*5=1	-4
2	10	0	2	0.2*10=2	-8
2	25	0	2	0.2*25=5 (**Q200**=2, 5>2, so the value 2 is used.)	-23
2	100	0	2	0.2*100=20 (**Q200**=2, 20>2, so the value 2 is used.)	-98
5	2	0	5	0.2*2=0.4	-1.6
5	5	0	5	0.2*5=1	-4
5	10	0	5	0.2*10=2	-8
5	25	0	5	0.2*25=5	-20
5	100	0	5	0.2*100=20 (**Q200**=5, 20>5, so the value 5 is used.)	-95
20	2	0	20	0.2*2=0.4	-1.6
20	5	0	20	0.2*5=1	-4
20	10	0	20	0.2*10=2	-8
20	25	0	20	0.2*25=5	-20
20	100	0	20	0.2*100=20	-80

Chip removal

The point at which the control removes chips also plays a decisive role for the work with overlong tools. The retraction position during the chip removal process does not have to be at the start position for drilling. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

- The chips are removed when the tool is positioned at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.

STARTING POINT Q379>0

Chip removal is at a certain value above the deepened starting point **Q379**. This value can be calculated as follows: **0.8 x Q379**; if the result of this calculation is larger than **Q200**, the value is always **Q200**.

Example:

- **SURFACE COORDINATE Q203** =0
- **SET-UP CLEARANCEQ200** =2
- **STARTING POINT Q379** =2

The position for chip removal is calculated as follows: 0.8 x **Q379**=0.8*2=1.6; the position for chip removal is 1.6 mm or inches above the deepened start point. So if the deepened starting point is at –2, the control starts chip removal at –0.4.

The following table shows examples of how the position for chip removal (retraction position) is calculated:

Position for chip removal (retraction position) with deepened starting point

Q200	Q379	Q203	Position at which pre-positioning is executed with FMAX	Factor 0.8 * Q379	Return position
2	2	0	2	0.8*2=1.6	-0.4
2	5	0	2	0.8*5=4	-3
2	10	0	2	0.8*10=8 (**Q200**=2, 8>2, so the value 2 is used.)	-8
2	25	0	2	0.8*25=20 (**Q200**=2, 20>2, so the value 2 is used.)	-23
2	100	0	2	0.8*100=80 (**Q200**=2, 80>2, so the value 2 is used.)	-98
5	2	0	5	0.8*2=1.6	-0.4
5	5	0	5	0.8*5=4	-1
5	10	0	5	0.8*10=8 (**Q200**=5, 8>5, so the value 5 is used.)	-5
5	25	0	5	0.8*25=20 (**Q200**=5, 20>5, so the value 5 is used.)	-20
5	100	0	5	0.8*100=80 (**Q200**=5, 80>5, so the value 5 is used.)	-95
20	2	0	20	0.8*2=1.6	-1.6
20	5	0	20	0.8*5=4	-4
20	10	0	20	0.8*10=8	-8
20	25	0	20	0.8*25=20	-20
20	100	0	20	0.8*100=80 (**Q200**=20, 80>20, so the value 20 is used.)	-80

14.3.6 Cycle 240 CENTERING

Application

Use Cycle **240 CENTERING** to machine center holes. You can specify the centering diameter or depth and an optional dwell time at the bottom. This dwell time is used for chip breaking at the bottom of the hole. If there is already a pilot hole then you can enter a deepened starting point.

Cycle sequence

1. From the current position, the control positions the tool at rapid traverse **FMAX** in the working plane to the starting position.
2. The control positions the tool at rapid traverse **FMAX** in the tool axis to the set-up clearance **Q200** above the workpiece surface **Q203**.
3. If you define **Q342 ROUGHING DIAMETER** not equal to 0, the control uses this value and the point angle of the tool **T-ANGLE** to calculate a deepened starting point. The control positions the tool at the **F PRE-POSITIONING Q253** feed rate to the deepened starting point.
4. The tool is centered at the programmed feed rate for plunging **F** to the programmed centering diameter or centering depth.
5. If a dwell time **Q211** is defined, the tool remains at the centering depth.
6. Finally, the tool is retracted to the set-up clearance or to the 2nd set-up clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**.

Notes

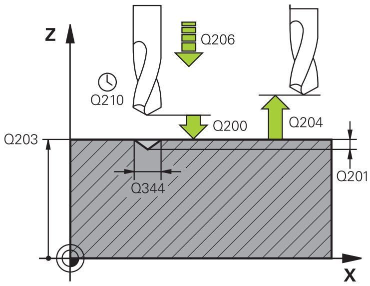
NOTICE
Danger of collision!
If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!
▶ Enter depth as negative ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.

Notes on programming

- Program a positioning block to position the tool at the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the **Q344** (diameter) or **Q201** (depth) cycle parameter determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q343 Select diameter/depth (1/0) Select whether centering is based on the entered diameter or depth. If the control is to center based on the entered diameter, the point angle of the tool must be defined in the **T-ANGLE** column of the TOOL.T tool table. **0**: Centering based on the entered depth **1**: Centering based on the entered diameter Input: **0**, **1**
	Q201 Depth? Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if **Q343**=0 is defined. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q344 Diameter of counterbore Centering diameter. Only effective if **Q343**=1 is defined. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while centering Input: **0...99999.999** or **FAUTO**, **FU**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q342 Roughing diameter? **0**: There is no hole **>0**: Diameter of the pre-drilled hole Input: **0...99999.9999**

Help graphic	Parameter
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the deepened starting point. The speed is in mm/min. Only in effect if **Q342 ROUGHING DIAMETER** is not 0. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Example

```
11 CYCL DEF 240 CENTERING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q343=+1                 ;SELECT DIA./DEPTH ~
    Q201=-2                 ;DEPTH ~
    Q344=-10                ;DIAMETER ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q211=+0                 ;DWELL TIME AT DEPTH ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q342=+12                ;ROUGHING DIAMETER ~
    Q253=+500               ;F PRE-POSITIONING
12 L  X+30  Y+20 R0 FMAX M3 M99
13 L  X+80  Y+50 R0 FMAX M99
```

14.3.7 Cycle 206 TAPPING

Application

The thread is cut in one or more passes. A floating tap holder is used.

Cycle sequence

1. The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2. The tool drills to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to set-up clearance at the end of the dwell time. If programmed, the tool moves to 2nd set-up clearance at **FMAX**
4. At the set-up clearance, the direction of spindle rotation reverses once again.

> A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4**.
- In Cycle **206**, the control uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

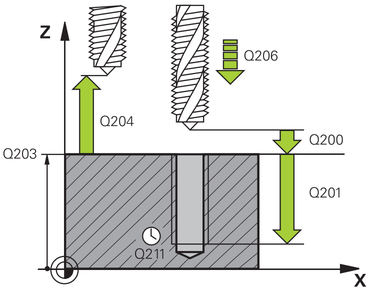
Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603):
    FeedPotentiometer (**default**) (speed override is not active), the control then adjusts the speed as required
    SpindlePotentiometer (feed rate override is not active)
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Guide value: 4 times the thread pitch Input: **0...99999.9999** or **PREDEF**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool during tapping Input: **0...99999.999** or **FAUTO**
	Q211 Dwell time at the depth? Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input: **0...3600.0000** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 CYCL DEF 206 TAPPING ~
    Q200=+2        ;SET-UP CLEARANCE ~
    Q201=-18       ;DEPTH OF THREAD ~
    Q206=+150      ;FEED RATE FOR PLNGNG ~
    Q211=+0        ;DWELL TIME AT DEPTH ~
    Q203=+0        ;SURFACE COORDINATE ~
    Q204=+50       ;2ND SET-UP CLEARANCE
12 CYCL CALL
```

The feed rate is calculated as follows: F = S x p

F: Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)

Retracting after a program interruption

Retracting in the Program Run, Single Block or Full Sequence mode

- To interrupt the program, select the **NC stop** key

Manual traverse

- Select **MANUAL TRAVERSE**
- Retract the tool in the active tool axis

Approach position

- To resume program execution, select **RESTORE POSITION**
- > A window is opened where the control shows the axis sequence as well as target position, current position and distance-to-go.
- Select the **NC start** key
- > The control moves the tool to the depth where it was stopped.
- To resume program execution, select **NC start** again

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

14.3.8 Cycle 207 RIGID TAPPING

Application

Refer to your machine manual.
Machine and control must be specially prepared by the machine manufacturer for use of this cycle.
This cycle is effective only for machines with servo-controlled spindle.

The control cuts the thread without a floating tap holder in one or more passes.

Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2 The tool drills to the total hole depth in one movement.
3 It then reverses the direction of spindle rotation and the tool is retracted to set-up clearance. If programmed, the tool moves to 2nd set-up clearance at **FMAX**
4 The control stops the spindle turning at that set-up clearance

For tapping, the spindle and the tool axis are always synchronized with each other. The synchronization can be carried out while the spindle is rotating or while it is stationary.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program **M3** (or **M4**) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the **TOOL CALL** block).
- If you do not program **M3** (or **M4**) before this cycle, the spindle will stand still after the end of the cycle. In this case, you must restart the spindle with **M3** (or **M4**) before the next operation.
- If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

If you do not change any dynamic parameters (e.g. set-up clearance, spindle speed,...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis leaves the acceleration path within this distance.

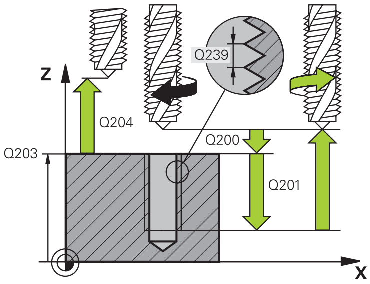
Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (spindle speed override is not active); the control then adjusts the spindle speed as required
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
  - **limitSpindleSpeed** (no. 113604): Spindle speed limit
    True: At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time.
    False: Limiting not active

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 CYCL DEF 207 RIGID TAPPING ~
   Q200=+2                 ;SET-UP CLEARANCE ~
   Q201=-18                ;DEPTH OF THREAD ~
   Q239=+1                 ;THREAD PITCH ~
   Q203=+0                 ;SURFACE COORDINATE ~
   Q204=+50                ;2ND SET-UP CLEARANCE
12 CYCL CALL
```

Retracting after a program interruption

Retracting in the Program Run, Single Block or Full Sequence mode

- To interrupt the program, select the **NC stop** key

Manual traverse

- Select **MANUAL TRAVERSE**
- Retract the tool in the active tool axis

Approach position

- To resume program execution, select **RESTORE POSITION**
- > A window is opened where the control shows the axis sequence as well as target position, current position and distance-to-go.
- Select the **NC start** key
- > The control moves the tool to the depth where it was stopped.
- To resume program execution, select **NC start** again

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

14.3.9 Cycle 209 TAPPING W/ CHIP BRKG

Application

Refer to your machine manual.
Machine and control must be specially prepared by the machine manufacturer for use of this cycle.
This cycle is effective only for machines with servo-controlled spindle.

The tool machines the thread in several passes until it reaches the programmed depth. You can define in a parameter whether the tool is to be retracted completely from the hole for chip breaking.

Cycle sequence

1 The control positions the tool in the tool axis at rapid traverse **FMAX** to the programmed set-up clearance above the workpiece surface.There, it carries out an oriented spindle stop
2 The tool moves to the programmed infeed depth, reverses the direction of spindle rotation and retracts by a specific distance or completely for chip release, depending on the definition. If you have defined a factor for increasing the spindle speed, the control retracts from the hole at the corresponding speed
3 It then reverses the direction of spindle rotation again and advances to the next infeed depth.
4 The control repeats this procedure (steps 2 to 3) until the programmed thread depth is reached
5 The tool is then retracted to set-up clearance. If programmed, the tool moves to 2nd set-up clearance at **FMAX**
6 The control stops the spindle turning at that set-up clearance

> For tapping, the spindle and the tool axis are always synchronized with each other. Synchronization may take place while the spindle is stationary.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program **M3** (or **M4**) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the **TOOL CALL** block).
- If you do not program **M3** (or **M4**) before this cycle, the spindle will stand still after the end of the cycle. In this case, you must restart the spindle with **M3** (or **M4**) before the next operation.
- If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

> If you do not change any dynamic parameters (e.g. set-up clearance, spindle speed,...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis leaves the acceleration path within this distance.

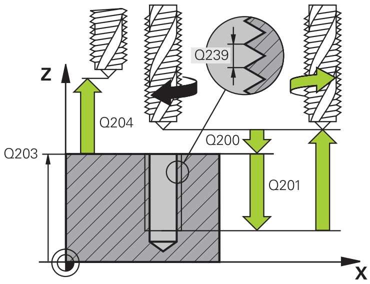
Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the cycle parameter "thread depth" determines the working direction.
- If you defined a speed factor for fast retraction in cycle parameter **Q403**, the control limits the speed to the maximum speed of the active gear stage.

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603):
    FeedPotentiometer (**default**) (speed override is not active), the control then adjusts the speed as required
    SpindlePotentiometer (feed rate override is not active)
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

Cycle parameters

Help graphic	Parameter
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q257 Infeed depth for chip breaking? Incremental depth at which the control performs chip breaking. This procedure is repeated until **DEPTH Q201** is reached. If **Q257** equals 0, the control will not perform chip breaking. This value has an incremental effect. Input: **0...99999.9999**
	Q256 Retract dist. for chip breaking? The control multiplies the pitch **Q239** by the programmed value and retracts the tool by the calculated value during chip breaking. If you enter **Q256** = 0, the control retracts the tool completely from the hole (to set-up clearance) for chip breaking. Input: **0...99999.9999**
	Q336 Angle for spindle orientation? Angle to which the control positions the tool before machining the thread. This allows you to re-cut the thread, if required. The value has an absolute effect. Input: **0...360**

Help graphic	Parameter
	Q403 RPM factor for retraction? Factor by which the control increases the spindle speed—and therefore also the retraction feed rate—when retracting from the drill hole. Maximum increase to maximum speed of the active gear stage. Input: **0.0001...10**

Example

```
11 CYCL DEF 209 TAPPING W/ CHIP BRKG ~
   Q200=+2              ;SET-UP CLEARANCE ~
   Q201=-18             ;DEPTH OF THREAD ~
   Q239=+1              ;THREAD PITCH ~
   Q203=+0              ;SURFACE COORDINATE ~
   Q204=+50             ;2ND SET-UP CLEARANCE ~
   Q257=+0              ;DEPTH FOR CHIP BRKNG ~
   Q256=+1              ;DIST FOR CHIP BRKNG ~
   Q336=+0              ;ANGLE OF SPINDLE ~
   Q403=+1              ;RPM FACTOR
12 CYCL CALL
```

Retracting after a program interruption

Retracting in the Program Run, Single Block or Full Sequence mode

- To interrupt the program, select the **NC stop** key
- Select **MANUAL TRAVERSE**
- Retract the tool in the active tool axis
- To resume program execution, select **RESTORE POSITION**
- > A window is opened where the control shows the axis sequence as well as target position, current position and distance-to-go.
- Select the **NC start** key
- > The control moves the tool to the depth where it was stopped.
- To resume program execution, select **NC start** again

NOTICE

Danger of collision!

If you move the tool in the negative direction instead of the positive direction when retracting it, there is a danger of collision.

- When retracting the tool you can move it in the positive and negative tool axis directions
- Be aware of the direction in which you retract the tool from the hole before retracting

14.3.10 Fundamentals of thread milling

Requirements

- Your machine tool features internal spindle cooling (cooling lubricant at least 30 bars, compressed air supply at least 6 bars)
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer (you can set the compensation in **TOOL CALL** using the **DR** delta radius).
- If you are using a left-cutting tool (**M4**), the type of milling in **Q351** is reversed
- The working direction is determined by the following input parameters: Algebraic sign **Q239** (+ = right-hand thread / – = left-hand thread) and type of milling **Q351** (+1 = climb / –1 = up-cut).

  The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

Internal thread	Pitch	Climb/Up-cut	Work direction
Right-handed	+	+1(RL)	Z+
Left-handed	–	–1(RR)	Z+
Right-handed	+	–1(RR)	Z–
Left-handed	–	+1(RL)	Z–

External thread	Pitch	Climb/Up-cut	Work direction
Right-handed	+	+1(RL)	Z–
Left-handed	–	–1(RR)	Z–
Right-handed	+	–1(RR)	Z+
Left-handed	–	+1(RL)	Z+

NOTICE

Danger of collision!

If you program the plunging depth values with different algebraic signs a collision may occur.

- Make sure to program all depth values with the same algebraic sign. Example: If you program the **Q356** COUNTERSINKING DEPTH parameter with a negative sign, then **Q201** DEPTH OF THREAD must also have a negative sign
- If you want to repeat just the counterbore procedure in a cycle, you can enter 0 for DEPTH OF THREAD. In this case, the machining direction is determined by the programmed COUNTERSINKING DEPTH

NOTICE

Danger of collision!

A collision may occur if, upon tool breakage, you retract the tool from the hole in the direction of the tool axis only.

- Stop the program run if the tool breaks
- Switch to the **Manual operation** operating mode in the **MDI** application
- First move the tool in a linear movement towards the hole center
- Retract the tool in the tool axis direction

> Programming and operating notes:
> - The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle **8 MIRRORING** in only one axis.
> - The programmed feed rate for thread milling references the cutting edge of the tool. However, since the control always displays the feed rate relative to the center path of the tool tip, the displayed value does not match the programmed value.

14.3.11 Cycle 262 THREAD MILLING

Application

With this cycle, you can mill a thread into pre-drilled material.

Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
2 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
3 The tool then approaches the nominal thread diameter tangentially in a helical movement. Before the helical approach, a compensating movement of the tool axis is carried out in order to begin at the programmed starting plane for the thread path
4 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.
5 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
6 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

> The nominal thread diameter is approached in a semi-circle from the center. A pre-positioning movement to the side is carried out if the tool diameter is smaller than the nominal thread diameter by four times the thread pitch.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

In the thread milling cycle, the tool will make a compensation movement in the tool axis before the approach. The length of the compensation movement is at most half of the thread pitch. This can result in a collision.

- Ensure sufficient space in the hole!

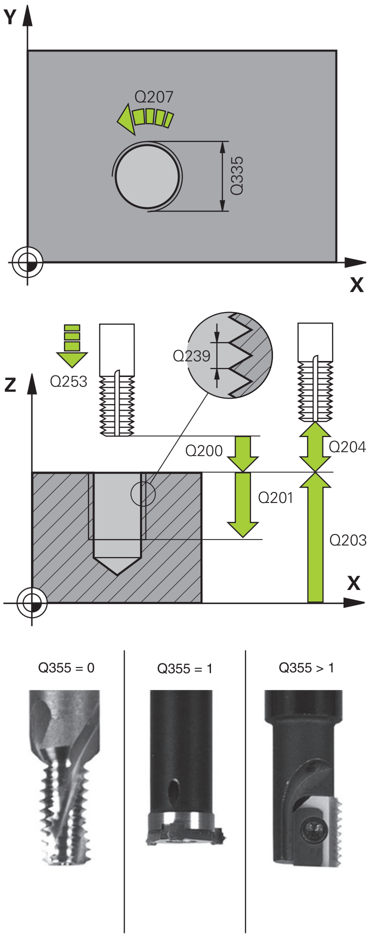
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you change the thread depth, the control will automatically move the starting point for the helical movement.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- If you program the thread depth =0, the cycle will not be executed.

Cycle parameters

Help graphic



Parameter

Q335 Nominal diameter?
Nominal thread diameter
Input: **0...99999.9999**

Q239 Pitch?
Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
+= right-hand thread
– = left-hand thread
Input: **-99.9999...+99.9999**

Q201 Depth of thread?
Distance between workpiece surface and root of thread. This value has an incremental effect.
Input: **-99999.9999...+99999.9999**

Q355 Number of threads per step?
Number of thread revolutions by which the tool is moved:
0 = one helical line to the thread depth
1 = continuous helical path over the entire length of the thread
>1 = several helical paths with approach and departure; between them, the control offsets the tool by **Q355**, multiplied by the pitch.
Input: **0...99999**

Q253 Feed rate for pre-positioning?
Traversing speed of the tool in mm/min when plunging or when retracting.
Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Q351 Direction? Climb=+1, Up-cut=-1
Type of milling operation. The direction of spindle rotation is taken into account.
+1 = climb milling
–1 = up-cut milling
(if you enter 0, climb milling is performed)
Input: **-1**, **0**, **+1** or **PREDEF**

Q200 Set-up clearance?
Distance between tool tip and workpiece surface. This value has an incremental effect.
Input: **0...99999.9999** or **PREDEF**

Q203 Workpiece surface coordinate?
Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**
	Q512 Feed rate for approaching? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 262 THREAD MILLING ~
   Q335=+5            ;NOMINAL DIAMETER ~
   Q239=+1            ;THREAD PITCH ~
   Q201=-18           ;DEPTH OF THREAD ~
   Q355=+0            ;THREADS PER STEP ~
   Q253=+750          ;F PRE-POSITIONING ~
   Q351=+1            ;CLIMB OR UP-CUT ~
   Q200=+2            ;SET-UP CLEARANCE ~
   Q203=+0            ;SURFACE COORDINATE ~
   Q204=+50           ;2ND SET-UP CLEARANCE ~
   Q207=+500          ;FEED RATE MILLING ~
   Q512=+0            ;FEED FOR APPROACH
12 CYCL CALL
```

14.3.12 Cycle 263 THREAD MLLNG/CNTSNKG

Application

With this cycle, you can mill a thread into pre-drilled material. In addition, you can use it to machine a countersunk chamfer.

Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

Countersinking

2 The tool moves at the feed rate for pre-positioning to the countersinking depth minus the set-up clearance, and then at the feed rate for countersinking to the countersinking depth.

3 If a set-up clearance to the side has been entered, the control immediately positions the tool at the pre-positioning feed rate to the countersinking depth.

4 Then, depending on the available space, the control smoothly approaches the tool to the core diameter, either tangentially from the center or with a pre-positioning movement to the side, and follows a circular path

Countersinking at front

5 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

6 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking

7 The tool then moves in a semicircle to the hole center

Thread milling

8 The control moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the algebraic sign of the thread pitch and the type of milling (climb or up-cut)

9 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion

10 After that the tool departs the contour tangentially and returns to the starting point in the working plane.

11 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

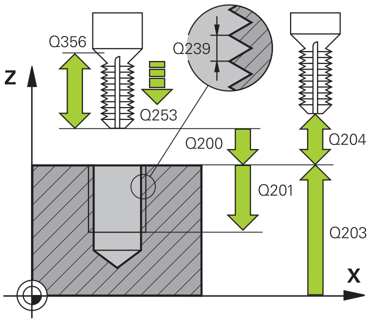
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
  1. Depth of thread
  2. Countersinking depth
  3. Depth at front

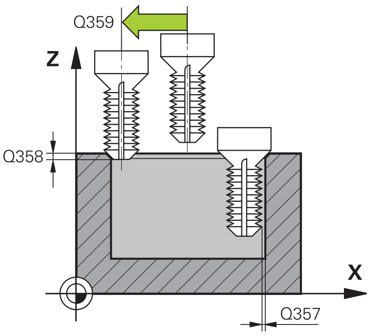
Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.
- If you want to countersink at front, define the countersinking depth as 0.

Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

Cycle parameters

Help graphic	Parameter
	**Q335 Nominal diameter?** Nominal thread diameter Input: **0...99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q356 Countersinking depth? Distance between tool point and the top surface of the workpiece. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (if you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	**Q357 Safety clearance to the side?** Distance between tool tooth and the wall. This value has an incremental effect. Input: **0...99999.9999**
	Q358 Sinking depth at front? Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q254 Feed rate for counterboring? Traversing speed of the tool in mm/min during counterboring Input: **0...99999.999** or **FAUTO**, **FU**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**
	Q512 Feed rate for approaching? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 263 THREAD MLLNG/CNTSNKG ~
    Q335=+5            ;NOMINAL DIAMETER ~
    Q239=+1            ;THREAD PITCH ~
    Q201=-18           ;DEPTH OF THREAD ~
    Q356=-20           ;COUNTERSINKING DEPTH ~
    Q253=+750          ;F PRE-POSITIONING ~
    Q351=+1            ;CLIMB OR UP-CUT ~
    Q200=+2            ;SET-UP CLEARANCE ~
    Q357=+0.2          ;CLEARANCE TO SIDE ~
    Q358=+0            ;DEPTH AT FRONT ~
    Q359=+0            ;OFFSET AT FRONT ~
    Q203=+0            ;SURFACE COORDINATE ~
    Q204=+50           ;2ND SET-UP CLEARANCE ~
    Q254=+200          ;F COUNTERBORING ~
    Q207=+500          ;FEED RATE MILLING ~
    Q512=+0            ;FEED FOR APPROACH
12 CYCL CALL
```

14.3.13 Cycle 264 THREAD DRILLNG/MLLNG

Application

With this cycle, you can drill into solid material, machine a counterbore, and finally mill a thread.

Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

Drilling

2 The tool drills to the first plunging depth at the programmed feed rate for plunging.
3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is retracted at rapid traverse to set-up clearance, and then moved again at **FMAX** to the entered advanced stop distance above the first plunging depth
4 The tool then advances with another infeed at the programmed feed rate.
5 The control repeats this procedure (steps 2 to 4) until the total drilling depth is reached

Countersinking at front

6 The tool moves at the feed rate for pre-positioning to the sinking depth at front.
7 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking
8 The tool then moves in a semicircle to the hole center

Thread milling

9 The control moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the algebraic sign of the thread pitch and the type of milling (climb or up-cut)
10 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion
11 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
12 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

Notes

NOTICE
Danger of collision! If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! ▶ Enter depth as negative ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The algebraic sign of the cycle parameters thread depth, countersinking depth or depth at front determines the working direction. The working direction is defined in the following sequence:
  1. Depth of thread
  2. Countersinking depth
  3. Depth at front

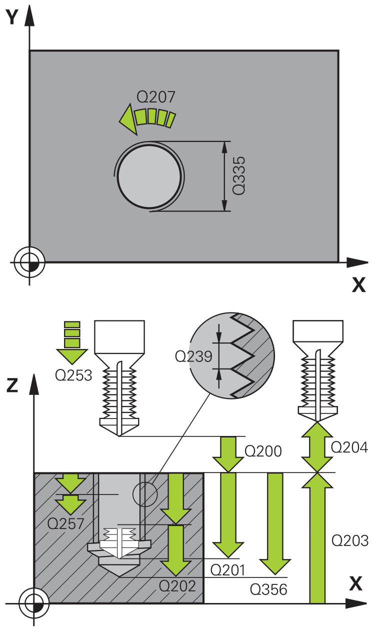
Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.

> Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

Cycle parameters

Help graphic



Parameter

Q335 Nominal diameter?

Nominal thread diameter

Input: **0...99999.9999**

Q239 Pitch?

Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

+= right-hand thread

– = left-hand thread

Input: **-99.9999...+99.9999**

Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q356 Total hole depth?

Distance between workpiece surface and hole bottom. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min when plunging or when retracting.

Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Q351 Direction? Climb=+1, Up-cut=-1

Type of milling operation. The direction of spindle rotation is taken into account.

+1 = climb milling

–1 = up-cut milling

(if you enter 0, climb milling is performed)

Input: **-1**, **0**, **+1** or **PREDEF**

Q202 Maximum plunging depth?

Infeed per cut. The **DEPTH Q201** does not have to be a multiple of **Q202**. This value has an incremental effect.

The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:

- the plunging depth is equal to the depth
- the plunging depth is greater than the depth

Input: **0...99999.9999**

Q258 Upper advanced stop distance?

Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after first chip removal. This value has an incremental effect.

Input: **0...99999.9999**

Help graphic	Parameter
	Q257 Infeed depth for chip breaking? Incremental depth at which the control performs chip breaking. This procedure is repeated until **DEPTH Q201** is reached. If **Q257** equals 0, the control will not perform chip breaking. This value has an incremental effect. Input: **0...99999.9999**
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0...99999.999** or **PREDEF**
	Q358 Sinking depth at front? Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. This value has an incremental effect. Input: **0...99999.9999**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q206 Feed rate for plunging? Tool traversing speed in mm/min during plunging Input: **0...99999.999** or **FAUTO**, **FU**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**
	Q512 Feed rate for approaching? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 264 THREAD DRILLNG/MLLNG ~
    Q335=+5              ;NOMINAL DIAMETER ~
    Q239=+1              ;THREAD PITCH ~
    Q201=-18             ;DEPTH OF THREAD ~
    Q356=-20             ;TOTAL HOLE DEPTH ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q351=+1              ;CLIMB OR UP-CUT ~
    Q202=+5              ;PLUNGING DEPTH ~
    Q258=+0.2            ;UPPER ADV STOP DIST ~
    Q257=+0              ;DEPTH FOR CHIP BRKNG ~
    Q256=+0.2            ;DIST FOR CHIP BRKNG ~
    Q358=+0              ;DEPTH AT FRONT ~
    Q359=+0              ;OFFSET AT FRONT ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q203=+0              ;SURFACE COORDINATE ~
    Q204=+50             ;2ND SET-UP CLEARANCE ~
    Q206=+150            ;FEED RATE FOR PLNGNG ~
    Q207=+500            ;FEED RATE MILLING ~
    Q512=+0              ;FEED FOR APPROACH
12 CYCL CALL
```

14.3.14 Cycle 265 HEL. THREAD DRLG/MLG

Application

With this cycle, you can mill a thread into solid material. In addition, you can choose to machine a counterbore before or after milling the thread.

Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

Countersinking at front

2 If countersinking occurs before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking occurs after thread milling, the control moves the tool to the countersinking depth at the feed rate for prepositioning
3 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking
4 The tool then moves in a semicircle to the hole center

Thread milling

5 The control moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread
6 The tool then approaches the nominal thread diameter tangentially in a helical movement
7 The tool moves on a continuous helical downward path until the thread depth value is reached
8 After that the tool departs the contour tangentially and returns to the starting point in the working plane.
9 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

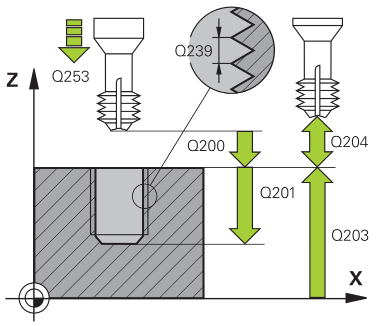
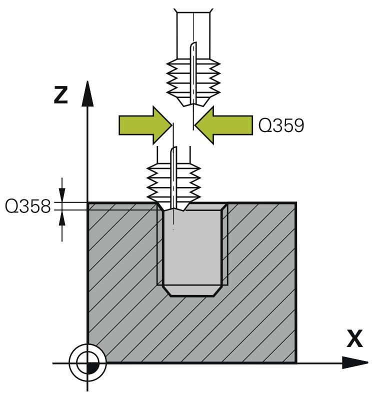
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you change the thread depth, the control will automatically move the starting point for the helical movement.
- The type of milling (up-cut or climb) is determined by the thread (right-hand or left-hand thread) and the direction of tool rotation, since it is only possible to work in the direction of the tool.
- The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
  1 Depth of thread
  2 Depth at front

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.

Cycle parameters

Help graphic	Parameter
	**Q335 Nominal diameter?** Nominal thread diameter Input: **0...99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	**Q358 Sinking depth at front?** Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. This value has an incremental effect. Input: **0...99999.9999**
	Q360 Countersink (before/after:0/1)? Execution of the chamfer **0** = before thread machining **1** = after thread machining Input: **0**, **1**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q254 Feed rate for counterboring? Traversing speed of the tool in mm/min during counterboring Input: **0...99999.999** or **FAUTO**, **FU**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 265 HEL. THREAD DRLG/MLG ~
    Q335=+5                ;NOMINAL DIAMETER ~
    Q239=+1                ;THREAD PITCH ~
    Q201=-18               ;DEPTH OF THREAD ~
    Q253=+750              ;F PRE-POSITIONING ~
    Q358=+0                ;DEPTH AT FRONT ~
    Q359=+0                ;OFFSET AT FRONT ~
    Q360=+0                ;COUNTERSINK PROCESS ~
    Q200=+2                ;SET-UP CLEARANCE ~
    Q203=+0                ;SURFACE COORDINATE ~
    Q204=+50               ;2ND SET-UP CLEARANCE ~
    Q254=+200              ;F COUNTERBORING ~
    Q207=+500              ;FEED RATE MILLING
12 CYCL CALL
```

14.3.15 Cycle 267 OUTSIDE THREAD MLLNG

Application

With this cycle, you can mill an external thread. In addition, you can use it to machine a countersunk chamfer.

Cycle sequence

1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface

Countersinking at front

2 The control approaches the starting point for countersinking at front, starting from the center of the stud, on the reference axis in the working plane. The position of the starting point is determined by the thread radius, tool radius and pitch

3 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

4 The control positions the tool without compensation from its center position on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking

5 The tool then moves on a semicircle to the starting point

Thread milling

6 The control positions the tool at the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front

7 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.

8 The tool then approaches the nominal thread diameter tangentially in a helical movement

9 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset helical movements or in one continuous helical movement.

10 After that the tool departs the contour tangentially and returns to the starting point in the working plane.

11 At the end of the cycle, the control retracts the tool at rapid traverse to setup clearance or—if programmed—to 2nd setup clearance

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

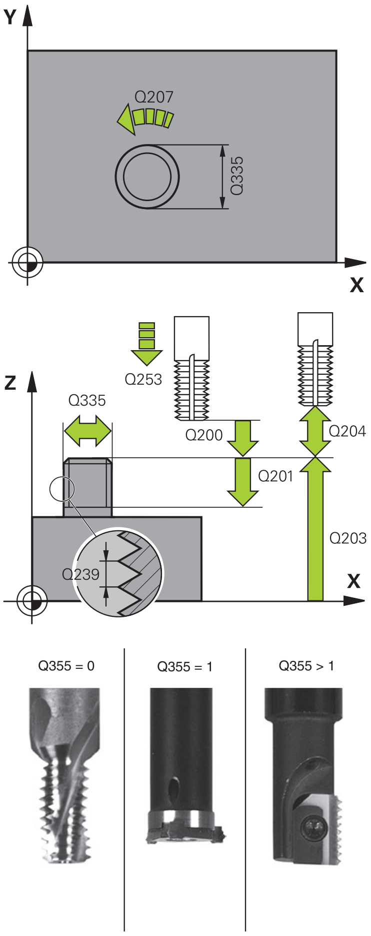
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).
- The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
  1. Depth of thread
  2. Depth at front

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- If you program one of the depth parameters to be 0, the control does not execute that step.

Cycle parameters

Help graphic	Parameter
	**Q335 Nominal diameter?** Nominal thread diameter Input: **0...99999.9999**
	Q239 Pitch? Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: **+**= right-hand thread **–** = left-hand thread Input: **-99.9999...+99.9999**
	Q201 Depth of thread? Distance between workpiece surface and root of thread. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q355 Number of threads per step? Number of thread revolutions by which the tool is moved: **0** = one helical line to the thread depth **1** = continuous helical path over the entire length of the thread **>1** = several helical paths with approach and departure; between them, the control offsets the tool by **Q355**, multiplied by the pitch. Input: **0...99999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when plunging or when retracting. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (if you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q358 Sinking depth at front? Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q359 Countersinking offset at front? Distance by which the control moves the tool center away from the center. This value has an incremental effect. Input: **0...99999.9999**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q254 Feed rate for counterboring? Traversing speed of the tool in mm/min during counterboring Input: **0...99999.999** or **FAUTO**, **FU**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min while milling Input: **0...99999.999** or **FAUTO**
	Q512 Feed rate for approaching? Traversing speed of the tool in mm/min while approaching. For smaller thread diameters, you can decrease the approaching feed rate in order to reduce the danger of tool breakage. Input: **0...99999.999** or **FAUTO**

Example

25 CYCL DEF 267 OUTSIDE THREAD MLLNG ~	
Q335=+10	;NOMINAL DIAMETER ~
Q239=+1.5	;THREAD PITCH ~
Q201=-20	;DEPTH OF THREAD ~
Q355=+0	;THREADS PER STEP ~
Q253=+750	;F PRE-POSITIONING ~
Q351=+1	;CLIMB OR UP-CUT ~
Q200=+2	;SET-UP CLEARANCE ~
Q358=+0	;DEPTH AT FRONT ~
Q359=+0	;OFFSET AT FRONT ~
Q203=+30	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q254=+150	;F COUNTERBORING ~
Q207=+500	;FEED RATE MILLING ~
Q512=+0	;FEED FOR APPROACH

14.3.16 Cycle 251 RECTANGULAR POCKET

Application

Use Cycle **251** to completely machine rectangular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Cycle sequence

Roughing

1 The tool plunges into the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with parameter **Q366**.
2 The control roughs out the pocket from the inside out, taking the path overlap (**Q370**) and the finishing allowances (**Q368** and **Q369**) into account.
3 At the end of the roughing operation, the control moves the tool tangentially away from the pocket wall, then moves to set-up clearance above the current plunging depth. From there, the tool is returned at rapid traverse to the pocket center.
4 This process is repeated until the programmed pocket depth is reached.

Finishing

5 If finishing allowances have been defined, the control plunges and then approaches the contour. The approach movement occurs on a radius in order to ensure a gentle approach. The control first finishes the pocket walls, with multiple infeeds, if so specified.
6 Then the control finishes the floor of the pocket from the inside out. The tool approaches the pocket floor tangentially

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

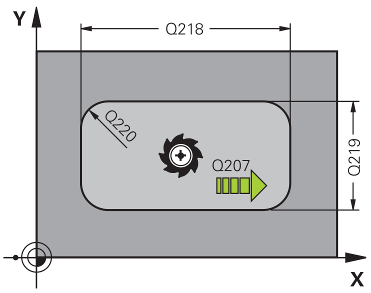
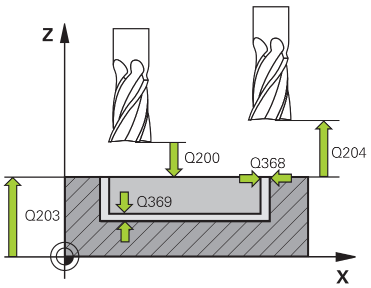
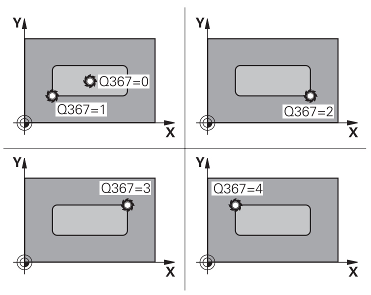
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- At the end, the control returns the tool to set-up clearance, or to 2nd set-up clearance if one was programmed.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- Cycle **251** takes the cutting width **RCUTS** from the tool table.

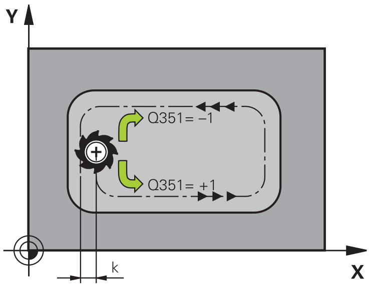
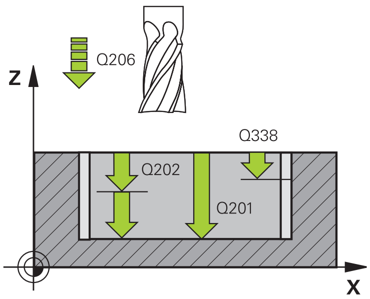
  Further information: "Plunging strategy Q366 with RCUTS", Page 518

Notes on programming

- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.
- Please note that you need to define sufficiently large workpiece blank dimensions if **Q224** Angle of rotation is not equal to 0.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	**Q218 First side length?** Pocket length, parallel to the main axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q219 Second side length? Pocket length, parallel to the secondary axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	**Q220 Corner radius?** Radius of the pocket corner. If you have entered 0 here, the control assumes that the corner radius is equal to the tool radius. Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	**Q224 Angle of rotation?** Angle by which the entire operation is rotated. The center of rotation is the position at which the tool is located when the cycle is called. The value has an absolute effect. Input: **-360.000...+360.000**
	Q367 Position of pocket (0/1/2/3/4)? Position of the pocket with respect to the tool when the cycle is called: **0**: Tool position = Center of pocket **1**: Tool position = Lower left corner **2**: Tool position = Lower right corner **3**: Tool position = Upper right corner **4**: Tool position = Upper left corner Input: **0**, **1**, **2**, **3**, **4**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Help graphic	Parameter
 	**Q351 Direction? Climb=+1, Up-cut=-1** Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q201 Depth? Distance between workpiece surface and bottom of pocket. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min for moving to depth Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.41** or **PREDEF**
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0**: Vertical plunging. The control plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table. **1**: Helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. If necessary, define the value of the **RCUTS** cutting width in the tool table **2**: Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. The reciprocation length depends on the plunging angle. As a minimum value, the control uses twice the tool diameter. If necessary, define the value of the **RCUTS** cutting width in the tool table **PREDEF**: The control uses the value from the GLOBAL DEF block Input: **0**, **1**, **2** or **PREDEF** Page 518
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 251 RECTANGULAR POCKET ~
    Q215=+0            ;MACHINING OPERATION ~
    Q218=+60           ;FIRST SIDE LENGTH ~
    Q219=+20           ;2ND SIDE LENGTH ~
    Q220=+0            ;CORNER RADIUS ~
    Q368=+0            ;ALLOWANCE FOR SIDE ~
    Q224=+0            ;ANGLE OF ROTATION ~
    Q367=+0            ;POCKET POSITION ~
    Q207=+500          ;FEED RATE MILLING ~
    Q351=+1            ;CLIMB OR UP-CUT ~
    Q201=-20           ;DEPTH ~
    Q202=+5            ;PLUNGING DEPTH ~
    Q369=+0            ;ALLOWANCE FOR FLOOR ~
    Q206=+150          ;FEED RATE FOR PLNGNG ~
    Q338=+0            ;INFEED FOR FINISHING ~
    Q200=+2            ;SET-UP CLEARANCE ~
    Q203=+0            ;SURFACE COORDINATE ~
    Q204=+50           ;2ND SET-UP CLEARANCE ~
    Q370=+1            ;TOOL PATH OVERLAP ~
    Q366=+1            ;PLUNGE ~
    Q385=+500          ;FINISHING FEED RATE ~
    Q439=+0            ;FEED RATE REFERENCE
12 L  X+50  Y+50 R0 FMAX M99
```

Plunging strategy Q366 with RCUTS

Helical plunging Q366 = 1

RCUTS > 0

- The control takes the cutting width **RCUTS** into account when calculating the helical path. The greater **RCUTS** is, the smaller the helical path.
- Formula for calculating the helical radius:

  $$Helicalradius = R_{corr} - RCUTS$$

  R_{corr}: Tool radius **R** + tool radius oversize **DR**
- If moving on a helical path is not possible due to limited space, the control will display an error message.

RCUTS = 0 or undefined

- The control does not monitor or modify the helical path.

Reciprocating plunge Q366 = 2

RCUTS > 0

- The control moves the tool along the complete reciprocating path.
- If moving on a reciprocating path is not possible due to limited space, the control will display an error message.

RCUTS = 0 or undefined

- The control moves the tool along one half of the reciprocating path.

14.3.17 Cycle 252 CIRCULAR POCKET

Application

Use Cycle **252** to machine circular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Cycle sequence

Roughing

1 The control first moves the tool at rapid traverse to set-up clearance **Q200** above the workpiece
2 The tool plunges to the first plunging depth at the pocket center. Specify the plunging strategy with parameter **Q366**.
3 The control roughs out the pocket from the inside out, taking the path overlap (**Q370**) and the finishing allowances (**Q368** and **Q369**) into account.
4 At the end of the roughing operation, the control moves the tool tangentially away from the pocket wall to set-up clearance **Q200** in the working plane, then retracts the tool by **Q200** at rapid traverse and returns it from there at rapid traverse to the pocket center
5 Steps 2 to 4 are repeated until the programmed pocket depth is reached, taking the finishing allowance **Q369** into account.
6 If only roughing was programmed (**Q215**=1), the tool moves away from the pocket wall tangentially by the set-up clearance **Q200**, then retracts at rapid traverse to the second set-up clearance **Q204** in the tool axis and returns at rapid traverse to the pocket center.

Finishing

1 If finishing allowances have been defined, the control first finishes the pocket walls, in multiple infeeds, if so specified.
2 The control positions the tool in the tool axis near the pocket wall at a distance corresponding to the finishing allowance **Q368** plus the set-up clearance **Q200**
3 The control roughs out the pocket from the inside out, until the diameter **Q223** is reached
4 Then, the control again positions the tool in the tool axis near the pocket wall at a distance corresponding to the finishing allowance **Q368** plus the set-up clearance **Q200** and repeats the finishing procedure for the side wall at the new depth
5 The control repeats this process until the programmed diameter is reached
6 After machining to the diameter **Q223**, the control retracts the tool tangentially by the finishing allowance **Q368** plus the set-up clearance **Q200** in the working plane, then retracts it at rapid traverse to set-up clearance **Q200** in the tool axis and returns it to the pocket center.
7 Next, the control moves the tool in the tool axis to the depth **Q201** and finishes the floor of the pocket from the inside out. The tool approaches the pocket floor tangentially.
8 The control repeats this process until the depth **Q201** plus **Q369** is reached.
9 Finally, the tool moves away from the pocket wall tangentially by the set-up clearance **Q200**, then retracts at rapid traverse to set-up clearance **Q200** in the tool axis and returns at rapid traverse to the pocket center.

Notes

NOTICE
Danger of collision! If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! ▶ Enter depth as negative ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE
Danger of collision! If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse. ▶ Conduct a roughing operation beforehand ▶ Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- Cycle **252** takes the cutting width **RCUTS** from the tool table.

  Further information: "Plunging strategy Q366 with RCUTS", Page 524

Notes on programming

- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position (circle center) with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.

Note regarding machine parameters

- For helical plunging, the control will display an error message if the internally calculated helix diameter is less than twice the tool diameter. If you are using a center-cut tool, you can switch this monitoring function off via the **suppress-PlungeErr** machine parameter (no. 201006).

Cycle parameters

Help graphic | **Parameter**

Q215 Machining operation (0/1/2)?

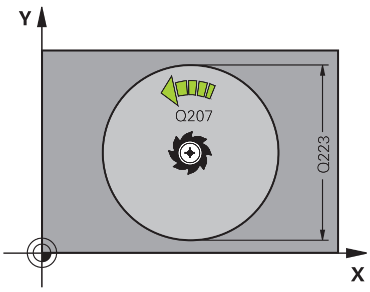
Define the machining operation:

0: Roughing and finishing

1: Only roughing

2: Only finishing
Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined

Input: **0**, **1**, **2**



Q223 Circle diameter?

Diameter of the finished pocket

Input: **0...99999.9999**

Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Q351 Direction? Climb=+1, Up-cut=-1

Type of milling operation. The direction of spindle rotation is taken into account.

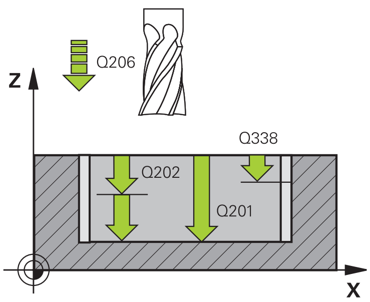
+1 = climb milling

-1 = up-cut milling

PREDEF: The control uses the value of a **GLOBAL DEF** block

(If you enter 0, climb milling is performed)

Input: **-1**, **0**, **+1** or **PREDEF**



Q201 Depth?

Distance between workpiece surface and bottom of pocket. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q202 Plunging depth?

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

Q369 Finishing allowance for floor?

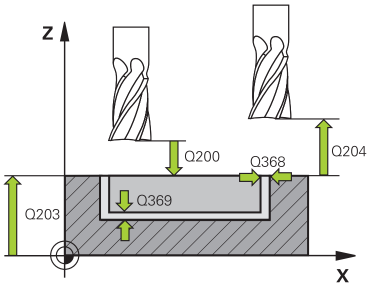
Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min for moving to depth

Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Help graphic	Parameter
	**Q338 Infeed for finishing?** Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q370 Path overlap factor? **Q370**x tool radius = stepover factor k. The overlap specified is the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input: **0.1...1999** or **PREDEF**
	Q366 Plunging strategy (0/1)? Type of plunging strategy: **0**: Vertical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as 0 or 90. Otherwise, the control will display an error message **1**: Helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. If necessary, define the value of the **RCUTS** cutting width in the tool table Input: **0**, **1** or **PREDEF** Page 524

Help graphic	Parameter
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 252 CIRCULAR POCKET ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q223=+50                ;CIRCLE DIAMETER ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q370=+1                 ;TOOL PATH OVERLAP ~
    Q366=+1                 ;PLUNGE ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q439=+0                 ;FEED RATE REFERENCE
12 L  X+50  Y+50 R0 FMAX M99
```

Plunging strategy Q366 with RCUTS

Behavior with RCUTS

Helical plunging **Q366=1**:

RCUTS > 0

- The control takes the cutting width **RCUTS** into account when calculating the helical path. The greater **RCUTS** is, the smaller the helical path.
- Formula for calculating the helical radius:

  $$Helicalradius = R_{corr} - RCUTS$$

  R_{corr}: Tool radius **R** + tool radius oversize **DR**
- If moving on a helical path is not possible due to limited space, the control will display an error message.

RCUTS = 0 or undefined

- **suppressPlungeErr**=**on** (no. 201006)

  If moving on a helical path is not possible due to limited space, the control will reduce the helical path.
- **suppressPlungeErr**=**off** (no. 201006)

  If moving on a helical radius is not possible due to limited space, the control will display an error message.

14.3.18 Cycle 253 SLOT MILLING

Application

Use Cycle **253** to completely machine a slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Cycle sequence

Roughing

1. Starting from the left slot arc center, the tool moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.
2. The control roughs out the slot from the inside out, taking the finishing allowances (**Q368** and **Q369**) into account
3. The control retracts the tool to set-up clearance **Q200**. If the slot width matches the cutter diameter, the control retracts the tool from the slot after each infeed
4. This process is repeated until the programmed slot depth is reached

Finishing

5. If a finishing allowance has been defined during pre-machining, the control first finishes the slot walls, using multiple infeeds, if so specified. The slot wall is approached tangentially in the left slot arc
6. Then the control finishes the floor of the slot from the inside out.

Notes

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start!

- Do **not** program any incremental dimensions after this cycle
- Program an absolute position in all main axes after this cycle

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

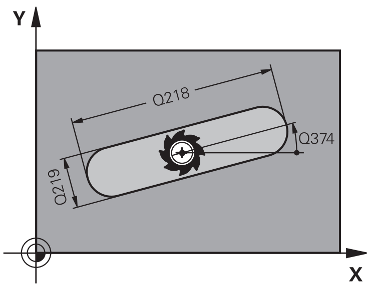
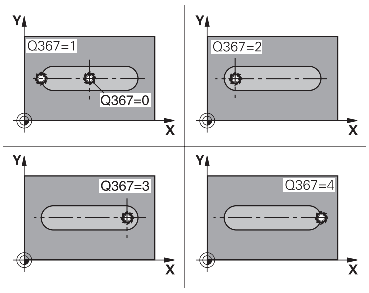
- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

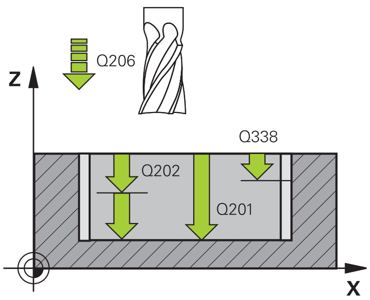
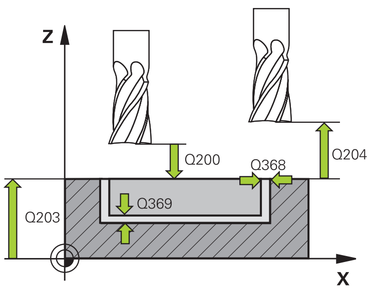
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- If the slot width is greater than twice the tool diameter, the control roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues an error message.

Notes on programming

- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	**Q218 Length of slot?** Enter the length of the slot. It is parallel to the main axis of the working plane. Input: **0...99999.9999**
	Q219 Width of slot? Enter the width of the slot. It is in parallel to the secondary axis of the working plane. If you enter a slot width that equals the tool diameter, then the control will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q374 Angle of rotation? Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called. The value has an absolute effect. Input: **-360.000...+360.000**
	**Q367 Position of slot (0/1/2/3/4)?** Position of the figure relative to the position of the tool when the cycle is called: **0**: Tool position = Center of figure **1**: Tool position = Left end of figure **2**: Tool position = Center of left figure arc **3**: Tool position = Center of right figure arc **4**: Tool position = Right end of figure Input: **0**, **1**, **2**, **3**, **4**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Help graphic	Parameter
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	**Q201 Depth?** Distance between workpiece surface and slot floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min for moving to depth Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0** = Vertical plunging. The plunging angle **ANGLE** in the tool table is not evaluated. **1, 2**= Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. Or: **PREDEF** Input: **0**, **1**, **2**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 253 SLOT MILLING ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q218=+60                 ;SLOT LENGTH ~
    Q219=+10                 ;SLOT WIDTH ~
    Q368=+0                  ;ALLOWANCE FOR SIDE ~
    Q374=+0                  ;ANGLE OF ROTATION ~
    Q367=+0                  ;SLOT POSITION ~
    Q207=+500                ;FEED RATE MILLING ~
    Q351=+1                  ;CLIMB OR UP-CUT ~
    Q201=-20                 ;DEPTH ~
    Q202=+5                  ;PLUNGING DEPTH ~
    Q369=+0                  ;ALLOWANCE FOR FLOOR ~
    Q206=+150                ;FEED RATE FOR PLNGNG ~
    Q338=+0                  ;INFEED FOR FINISHING ~
    Q200=+2                  ;SET-UP CLEARANCE ~
    Q203=+0                  ;SURFACE COORDINATE ~
    Q204=+50                 ;2ND SET-UP CLEARANCE ~
    Q366=+2                  ;PLUNGE ~
    Q385=+500                ;FINISHING FEED RATE ~
    Q439=+3                  ;FEED RATE REFERENCE
12 L  X+50  Y+50 R0 FMAX M99
```

14.3.19 Cycle 254 CIRCULAR SLOT

Application

Use Cycle **254** to completely machine a circular slot. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Cycle sequence

Roughing

1 The tool moves in a reciprocating motion in the slot center at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.
2 The control roughs out the slot from the inside out, taking the finishing allowances (**Q368** and **Q369**) into account
3 The control retracts the tool to set-up clearance **Q200**. If the slot width matches the cutter diameter, the control retracts the tool from the slot after each infeed
4 This process is repeated until the programmed slot depth is reached

Finishing

5 If finishing allowances have been defined, the control first finishes the slot walls, in multiple infeeds, if so specified. The slot wall is approached tangentially
6 Then the control finishes the floor of the slot from the inside out

Notes

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start!

- Do **not** program any incremental dimensions after this cycle
- Program an absolute position in all main axes after this cycle

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

- Conduct a roughing operation beforehand
- Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

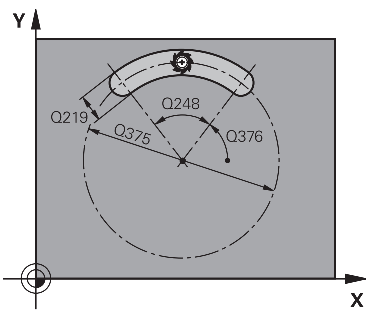
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.

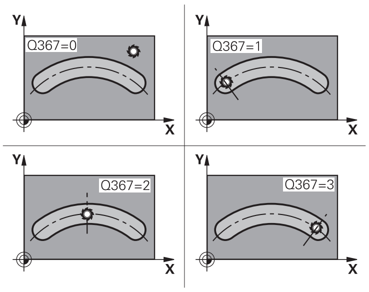
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- If the slot width is greater than twice the tool diameter, the control roughs the slot correspondingly from the inside out. You can therefore mill any slots with small tools, too.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- The control uses the **RCUTS** value in the cycle to monitor non-center-cut tools and to prevent the tool from front-face touching. If necessary, the control interrupts machining and issues an error message.

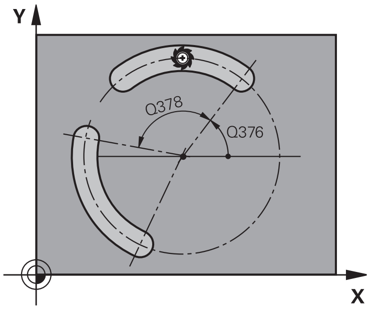
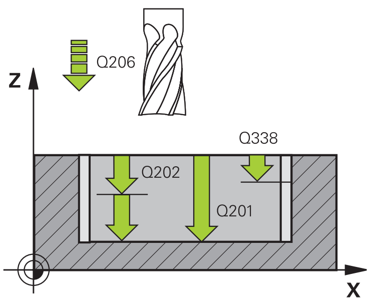
Notes on programming

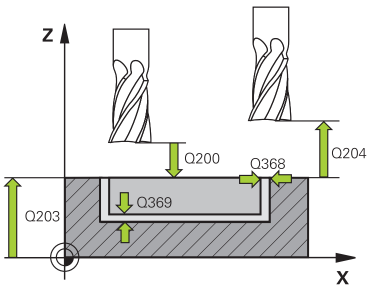
- If the tool table is inactive, you must always plunge vertically (**Q366**=0) because you cannot define a plunging angle.
- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.
- Slot position 0 is not allowed if you use Cycle **254** in combination with Cycle **221**.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	**Q219 Width of slot?** Enter the width of the slot. It is in parallel to the secondary axis of the working plane. If you enter a slot width that equals the tool diameter, then the control will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q375 Pitch circle diameter? Enter the diameter of the circle. Input: **0...99999.9999**

Help graphic	Parameter
	**Q367 Ref. for slot pos. (0/1/2/3)?** Position of the slot relative to the position of the tool when the cycle is called: **0**: The tool position is not taken into account. The slot position is determined from the entered pitch circle center and the starting angle. **1**: Tool position = Center of left slot circle. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account. **2**: Tool position = Center of center line. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account. **3**: Tool position = Center of right slot circle. Starting angle **Q376** refers to this position. The entered pitch circle center is not taken into account. Input: **0**, **1**, **2**, **3**
	Q216 Center in 1st axis? Center of the pitch circle in the main axis of the working plane. **Only effective if Q367 = 0**. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q217 Center in 2nd axis? Center of the pitch circle in the secondary axis of the working plane. **Only effective if Q367 = 0**. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	**Q376 Starting angle?** Enter the polar angle of the starting point. The value has an absolute effect. Input: **-360.000...+360.000**
	Q248 Angular length? Enter the angular length of the slot. This value has an incremental effect. Input: **0...360**
	Q378 Intermediate stepping angle? Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle. This value has an incremental effect. Input: **-360.000...+360.000**
	Q377 Number of repetitions? Number of machining operations on a pitch circle Input: **1...99999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	**Q201 Depth?** Distance between workpiece surface and slot floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min for moving to depth Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
 	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0**: Vertical plunging. The plunging angle (ANGLE) in the tool table is not evaluated. **1, 2**: Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message **PREDEF**: The control uses the value from the GLOBAL DEF block. Input: **0**, **1**, **2**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Help graphic	Parameter
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 254 CIRCULAR SLOT ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q219=+10                ;SLOT WIDTH ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q375=+60                ;PITCH CIRCLE DIAMETR ~
    Q367=+0                 ;REF. SLOT POSITION ~
    Q216=+50                ;CENTER IN 1ST AXIS ~
    Q217=+50                ;CENTER IN 2ND AXIS ~
    Q376=+0                 ;STARTING ANGLE ~
    Q248=+0                 ;ANGULAR LENGTH ~
    Q378=+0                 ;STEPPING ANGLE ~
    Q377=+1                 ;NR OF REPETITIONS ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q366=+2                 ;PLUNGE ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q439=+0                 ;FEED RATE REFERENCE
12 L  X+50  Y+50 R0 FMAX M99
```

14.3.20 Cycle 256 RECTANGULAR STUD

Application

Use Cycle **256** to machine a rectangular stud. If a dimension of the workpiece blank is greater than the maximum possible stepover, then the control performs multiple stepovers until the finished dimension has been machined.

Cycle sequence

1 The tool moves from the cycle starting position (stud center) to the starting position for stud machining. Specify the starting position with parameter **Q437**. The default position (**Q437**=0) is 2 mm to the right of the stud blank
2 If the tool is at the 2nd set-up clearance, it moves at rapid traverse **FMAX** to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging
3 The tool then moves tangentially to the stud contour and machines one revolution
4 If the finished dimension cannot be machined with one revolution, the control performs a stepover with the current factor, and machines another revolution. The control takes the dimensions of the workpiece blank, the finished dimension, and the permitted stepover into account. This process is repeated until the defined finished dimension has been reached. If, on the other hand, you did not set the starting point on a side, but rather on a corner (**Q437** not equal to 0), the control mills on a spiral path from the starting point inward until the finished dimension has been reached.
5 If further stepovers are required, the tool is retracted from the contour on a tangential path and returns to the starting point of stud machining
6 The control then plunges the tool to the next plunging depth, and machines the stud at this depth
7 This process is repeated until the programmed stud depth is reached
8 At the end of the cycle, the control positions the tool in the tool axis at the clearance height defined in the cycle. This means that the end position differs from the starting position

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

If there is not enough room for the approach movement next to the stud, there is danger of collision.

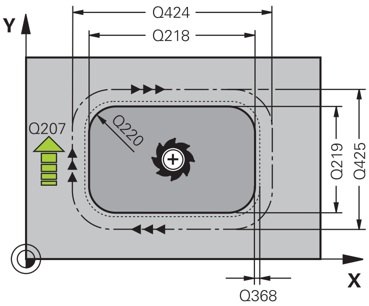
- Depending on the approach position **Q439**, leave enough room next to the stud for the approach movement
- Leave room next to the stud for the approach motion
- At least tool diameter + 2 mm
- At the end, the control returns the tool to set-up clearance, or to 2nd set-up clearance if one was programmed. The end position of the tool after the cycle differs from the starting position.

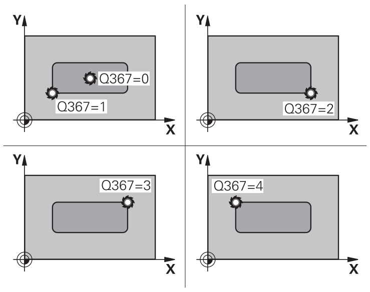
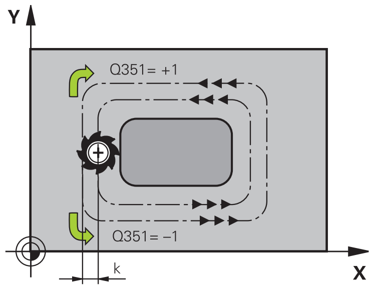
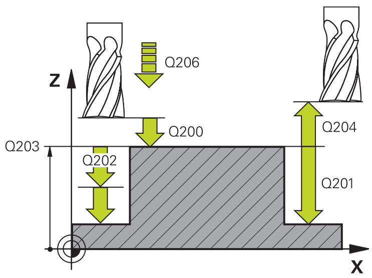
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
	**Q218 First side length?** Length of stud parallel to the main axis of the working plane Input: **0...99999.9999**
	Q424 Workpiece blank side length 1? Length of stud blank parallel to the main axis of the working plane. Enter **Workpiece blank side length 1** greater than **First side length**. The control performs multiple lateral stepovers if the difference between blank dimension 1 and finished dimension 1 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input: **0...99999.9999**
	Q219 Second side length? Length of stud parallel to the secondary axis of the working plane Enter **Workpiece blank side length 2** greater than **Second side length**. The control performs multiple lateral stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input: **0...99999.9999**
	Q425 Workpiece blank side length 2? Length of stud blank parallel to the secondary axis of the working plane. Input: **0...99999.9999**
	Q220 Radius / Chamfer (+/-)? Enter the value for the radius or chamfer form element. If you enter a positive value, the control will round every corner. The value you enter here refers to the radius. If you enter a negative value, all corners of the contour will be chamfered with the value entered as the length of the chamfer. Input: **-99999.9999...+99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane, is left over after machining. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q224 Angle of rotation? Angle by which the entire operation is rotated. The center of rotation is the position at which the tool is located when the cycle is called. The value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	**Q367 Position of stud (0/1/2/3/4)?** Position of the stud with respect to the tool when the cycle is called. **0**: Tool position = Center of stud **1**: Tool position = Lower left corner **2**: Tool position = Lower right corner **3**: Tool position = Upper right corner **4**: Tool position = Upper left corner Input: **0**, **1**, **2**, **3**, **4**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	**Q351 Direction? Climb=+1, Up-cut=-1** Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	**Q201 Depth?** Distance between workpiece surface and bottom of stud. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while moving to depth Input: **0...99999.999** or **FAUTO**, **FMAX**, **FU**, **FZ**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.9999** or **PREDEF**
	Q437 Starting position (0...4)? Specify the approach strategy of the tool: **0**: From the right of the stud (default setting) **1**: Lower left corner **2**: Lower right corner **3**: Upper right corner **4**: Upper left corner If approach marks appear on the stud surface during approach with the setting **Q437**=0, then choose another approach position. Input: **0**, **1**, **2**, **3**, **4**
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Example

```
11 CYCL DEF 256 RECTANGULAR STUD ~
    Q218=+60                ;FIRST SIDE LENGTH ~
    Q424=+75                ;WORKPC. BLANK SIDE 1 ~
    Q219=+20                ;2ND SIDE LENGTH ~
    Q425=+60                ;WORKPC. BLANK SIDE 2 ~
    Q220=+0                 ;CORNER RADIUS ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q224=+0                 ;ANGLE OF ROTATION ~
    Q367=+0                 ;STUD POSITION ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q206=+3000              ;FEED RATE FOR PLNGNG ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q370=+1                 ;TOOL PATH OVERLAP ~
    Q437=+0                 ;APPROACH POSITION ~
    Q215=+1                 ;MACHINING OPERATION ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q385=+500               ;FEED RATE FOR FINISHING
12 L  X+50  Y+50 R0 FMAX M99
```

14.3.21 Cycle 257 CIRCULAR STUD

Application

Use Cycle **257** to machine a circular stud. The control mills the circular stud with a helical infeed motion starting from the workpiece blank diameter.

Cycle sequence

1. If the current position of the tool is below the 2nd set-up clearance, the control then lifts it off and retracts it to the 2nd set-up clearance.
2. The tool moves from the stud center to the starting position for stud machining. With the polar angle, you specify the starting position with respect to the stud center using parameter **Q376**.
3. The control moves the tool at rapid traverse **FMAX** to set-up clearance **Q200**, and from there advances to the first plunging depth at the feed rate for plunging
4. The control then machines the circular stud with a helical infeed motion, taking the path overlap into account
5. The control retracts the tool from the contour by 2 mm on a tangential path
6. If more than one plunging movement is required, the tool repeats the plunging movement at the point next to the departure movement
7. This process is repeated until the programmed stud depth is reached
8. At the end of the cycle, the tool firsts departs on a tangential path and is then retracted in the tool axis to the 2nd set-up clearance defined in the cycle. This means that the end position differs from the starting position

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

There is a danger of collision if there is insufficient room next to the stud.

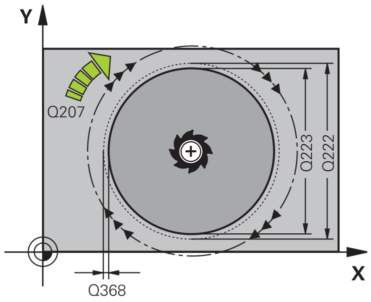
- Check the machining sequence using the graphic simulation.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

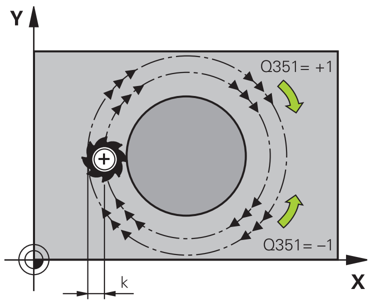
Notes on programming

- Pre-position the tool in the working plane to the starting position (stud center) with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

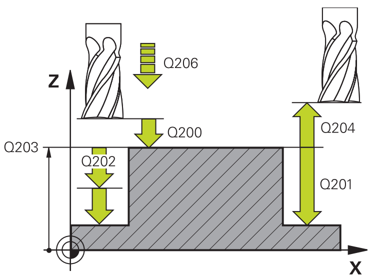
Cycle parameters



Help graphic	Parameter
	Q223 Finished part diameter? Diameter of the finished stud Input: **0...99999.9999**
	Q222 Workpiece blank diameter? Diameter of workpiece blank. The workpiece blank diameter must be greater than the diameter of the finished part. The control performs multiple stepovers if the difference between the workpiece blank diameter and reference circle diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**



Help graphic	Parameter
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**



Help graphic	Parameter
	Q201 Depth? Distance between workpiece surface and bottom of stud. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while moving to depth Input: **0...99999.999** or **FAUTO**, **FMAX**, **FU**, **FZ**

Help graphic	Parameter
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.9999** or **PREDEF**
	Q376 Starting angle? Polar angle relative to the stud center, from which the tool approaches the stud. Input: **-1...+359**
	Q215 Machining operation (0/1/2)? Specify the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Input: **0**, **1**, **2**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect.
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Example

```
11 CYCL DEF 257 CIRCULAR STUD ~
    Q223=+50                 ;FINISHED PART DIA. ~
    Q222=+52                 ;WORKPIECE BLANK DIA. ~
    Q368=+0                  ;ALLOWANCE FOR SIDE ~
    Q207=+500                ;FEED RATE MILLING ~
    Q351=+1                  ;CLIMB OR UP-CUT ~
    Q201=-20                 ;DEPTH ~
    Q202=+5                  ;PLUNGING DEPTH ~
    Q206=+3000               ;FEED RATE FOR PLNGNG ~
    Q200=+2                  ;SET-UP CLEARANCE ~
    Q203=+0                  ;SURFACE COORDINATE ~
    Q204=+50                 ;2ND SET-UP CLEARANCE ~
    Q370=+1                  ;TOOL PATH OVERLAP ~
    Q376=-1                  ;STARTING ANGLE ~
    Q215=+1                  ;MACHINING OPERATION ~
    Q369=+0                  ;ALLOWANCE FOR FLOOR ~
    Q338=+0                  ;INFEED FOR FINISHING ~
    Q385=+500                ;FINISHING FEED RATE
12 L  X+50  Y+50 R0 FMAX M99
```

14.3.22 Cycle 258 POLYGON STUD

Application

Use Cycle **258** to machine a regular polygon by machining the contour outside. The milling operation is carried out on a spiral path based on the diameter of the workpiece blank.

Cycle sequence

1. If, at the beginning of machining, the work piece is positioned below the 2nd set-up clearance, the control will retract the tool back to 2nd set-up clearance
2. Starting from the center of the stud the control moves the tool to the starting point of stud machining. The starting point depends, among other things, on the diameter of the workpiece blank and the angle of rotation of the stud. The angle of rotation is determined with parameter **Q224**
3. The tool moves at rapid traverse **FMAX** to the setup clearance **Q200** and from there with the feed rate for plunging to the first plunging depth.
4. The control then machines the circular stud with a helical infeed motion, taking the path overlap into account
5. The control moves the tool on a tangential path from the outside to the inside
6. The tool will be lifted in the direction of the spindle axis to 2nd set-up clearance in one rapid movement
7. If several plunging depths are required, the control returns the tool to the starting point of the stud milling process and then plunges the tool to the programmed depth
8. This process is repeated until the programmed stud depth is reached.
9. At the end of the cycle, first a departing motion is performed. Then the control will move the tool on the tool axis to 2nd set-up clearance

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

In this cycle, the control performs an automatic approach movement. If there is not enough space, a collision might occur.

- Use **Q224** to specify which angle is used to machine the first corner of the polygon stud. Input range: –360° to +360°
- Depending on the angle of rotation **Q224**, the following amount of space must be left next to the stud: At least tool diameter +2 mm

NOTICE

Danger of collision!

At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle need not be the same as the starting position.

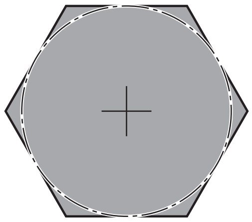
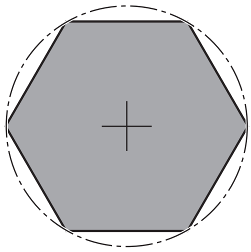
- Control the traversing movements of the machine
- In the **Simulation** workspace of the **Editor** operating mode, check the end position of the tool after the cycle.
- After the cycle, program the absolute (not incremental) coordinates

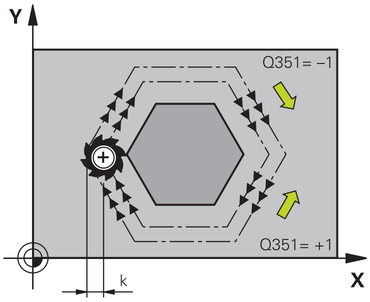
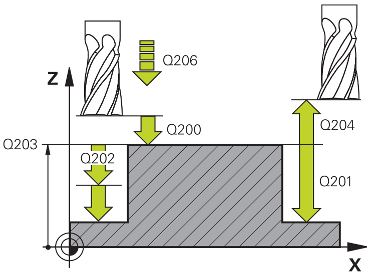
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Before the start of the cycle you will have to pre-position the tool in the working plane. In order to do so, move the tool with radius compensation **R0** to the center of the stud.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
Q573 = 0  Q573 = 1 	**Q573 Inscr.circle/circumcircle (0/1)?** Define whether the dimension **Q571** is referenced to the inscribed circle or the circumcircle: **0**: Dimension is referenced to the inscribed circle **1**: Dimension is referenced to the circumcircle Input: **0**, **1**
	Q571 Reference circle diameter? Enter the diameter of the reference circle. Specify in parameter **Q573** whether the diameter entered here is referenced to the inscribed circle or the circumcircle. You can program a tolerance if needed. Input: **0...99999.9999**
	Q222 Workpiece blank diameter? Enter the diameter of the blank. The workpiece blank diameter must be greater than the reference circle diameter. The control performs multiple stepovers if the difference between the workpiece blank diameter and reference circle diameter is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The control always calculates a constant stepover. Input: **0...99999.9999**
	Q572 Number of corners? Enter the number of corners of the polygon stud. The control distributes the corners evenly on the stud. Input: **3...30**
	Q224 Angle of rotation? Specify which angle is used to machine the first corner of the polygon stud. Input: **-360.000...+360.000**
	Q220 Radius / Chamfer (+/-)? Enter the value for the radius or chamfer form element. If you enter a positive value, the control will round every corner. The value you enter here refers to the radius. If you enter a negative value, all corners of the contour will be chamfered with the value entered as the length of the chamfer. Input: **-99999.9999...+99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. If you enter a negative value here, the control will return the tool to a diameter outside of the workpiece blank diameter after roughing. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Help graphic	Parameter
	**Q351 Direction? Climb=+1, Up-cut=-1** Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	**Q201 Depth?** Distance between workpiece surface and bottom of stud. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min while moving to depth Input: **0...99999.999** or **FAUTO**, **FMAX**, **FU**, **FZ**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.0001...1.9999** or **PREDEF**

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**

Example

```
11 CYCL DEF 258 POLYGON STUD ~
    Q573=+0                ;REFERENCE CIRCLE ~
    Q571=+50               ;REF-CIRCLE DIAMETER ~
    Q222=+52               ;WORKPIECE BLANK DIA. ~
    Q572=+6                ;NUMBER OF CORNERS ~
    Q224=+0                ;ANGLE OF ROTATION ~
    Q220=+0                ;RADIUS / CHAMFER ~
    Q368=+0                ;ALLOWANCE FOR SIDE ~
    Q207=+500              ;FEED RATE MILLING ~
    Q351=+1                ;CLIMB OR UP-CUT ~
    Q201=-20               ;DEPTH ~
    Q202=+5                ;PLUNGING DEPTH ~
    Q206=+3000             ;FEED RATE FOR PLNGNG ~
    Q200=+2                ;SET-UP CLEARANCE ~
    Q203=+0                ;SURFACE COORDINATE ~
    Q204=+50               ;2ND SET-UP CLEARANCE ~
    Q370=+1                ;TOOL PATH OVERLAP ~
    Q215=+0                ;MACHINING OPERATION ~
    Q369=+0                ;ALLOWANCE FOR FLOOR ~
    Q338=+0                ;INFEED FOR FINISHING ~
    Q385=+500              ;FINISHING FEED RATE
12 L  X+50  Y+50 R0 FMAX M99
```

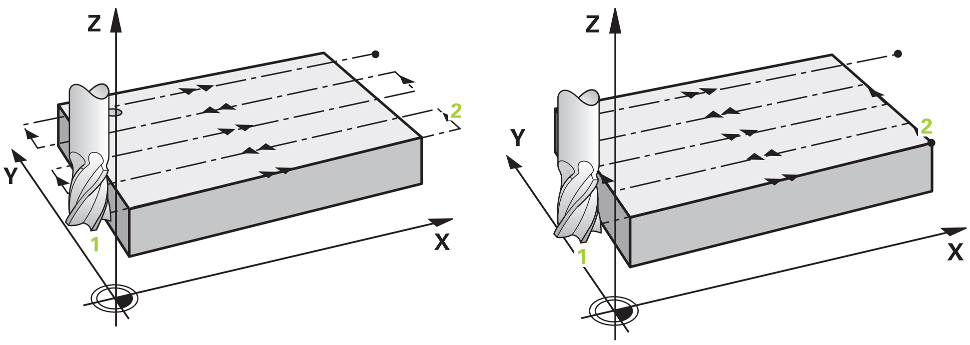
14.3.23 Cycle 233 FACE MILLING

Application

With Cycle **233**, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. You can also define side walls in the cycle, which are then taken into account when machining the level surface. The cycle offers you various machining strategies:

- **Strategy Q389=0**: Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: The surface is machined line by line with overtravel; stepover when retracting at rapid traverse
- **Strategy Q389=3**: The surface is machined line by line without overtravel; stepover when retracting at rapid traverse
- **Strategy Q389=4**: Helical machining from the outside toward the inside

Strategies Q389=0 and Q389 =1

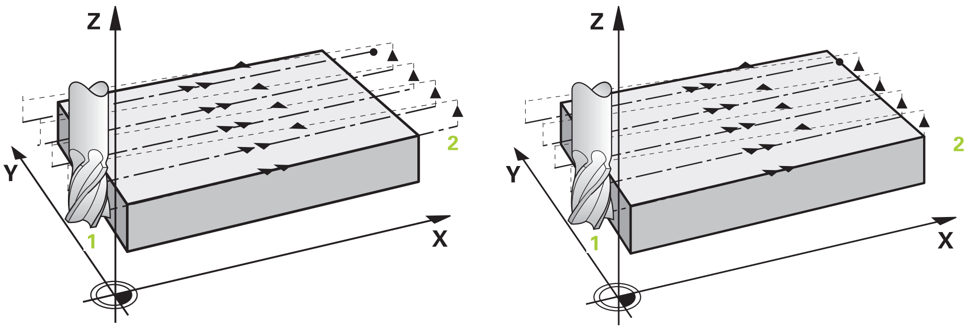


The strategies **Q389**=0 and **Q389**=1 differ in the overtravel during face milling. If **Q389**=0, the end point lies outside of the surface, with **Q389**=1, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy **Q389**=0 is used, the control additionally moves the tool beyond the level surface by the tool radius.

Cycle sequence

1. From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane. The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.
2. The control then positions the tool at rapid traverse **FMAX** to set-up clearance in the spindle axis.
3. The tool then moves in the spindle axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control.
4. The control moves the tool to end point **2** at the programmed feed rate for milling.
5. The control then shifts the tool laterally to the starting point of the next line at the pre-positioning feed rate. The control calculates the offset from the programmed width, the tool radius, the maximum path overlap factor and the set-up clearance to the side.
6. The tool then returns in the opposite direction at the feed rate for milling.
7. The process is repeated until the programmed surface has been machined completely.
8. The control then positions the tool at rapid traverse **FMAX** back to starting point **1**.
9. If more than one infeed is required, the control moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.
10. The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
11. At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**.

Strategies Q389=2 and Q389 =3



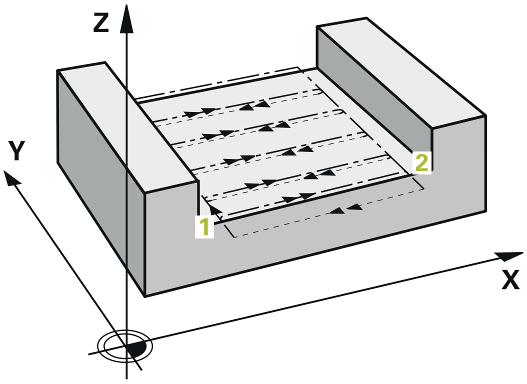
The strategies **Q389**=2 and **Q389**=3 differ in the overtravel during face milling. If **Q389**=2, the end point lies outside of the surface, with **Q389**=3, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy **Q389**=2 is used, the control additionally moves the tool beyond the level surface by the tool radius.

Cycle sequence

1 From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane. The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.
2 The control then positions the tool at rapid traverse **FMAX** to set-up clearance in the spindle axis.
3 The tool then moves in the spindle axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control.
4 The tool subsequently advances at the programmed feed rate for milling to the end point **2**.
5 The control positions the tool in the tool axis to the set-up clearance above the current infeed depth, and then moves at **FMAX** directly back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius, the maximum path overlap factor and the set-up clearance to the side.
6 The tool then returns to the current infeed depth and moves in the direction of the end point **2**.
7 The process is repeated until the programmed surface has been machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**.
8 If more than one infeed is required, the control moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.
9 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
10 At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**.

Strategies Q389=2 and Q389=3

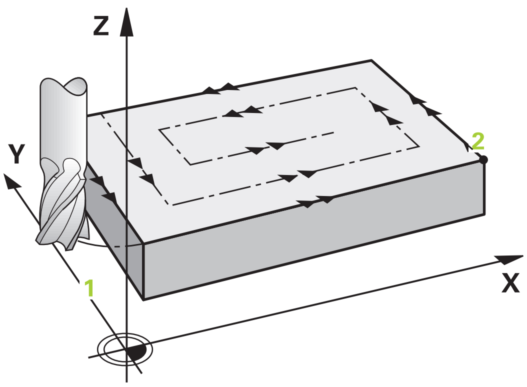
If you program a lateral limitation, the control might not be able to perform movements outside of the contour. In this case the cycle runs as follows:



1 The control positions the tool at **FMAX** to the starting point in the working plane. This position is offset from the edge of the workpiece by the tool radius and the set-up clearance **Q357** to the side.
2 The tool traverses to the set-up clearance **Q200** at rapid traverse **FMAX** in the tool axis and from there to the first plunging depth **Q202** at **Q207 FEED RATE MILLING**.
3 The control moves the tool on a circular path to the starting point point **1**.
4 The tool moves at the programmed feed rate **Q207** to the end point **2** and departs from the contour on a circular path.
5 Then the control moves the tool to the approach position of the next path at **Q253 F PRE-POSITIONING** .
6 Steps 3 to 5 are repeated until the entire surface is milled.
7 If more than one infeed depth is programmed, the control moves the tool at the end of the last path to the set-up clearance **Q200** and positions in the working plane to the next approach position.
8 In the last infeed, the control mills **Q369 ALLOWANCE FOR FLOOR** at **Q385 FINISHING FEED RATE**.
9 In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
10 At the end of the last path, the control retracts the tool to the 2nd set-up clearance **Q204** and then to the position last programmed before the cycle.

- The circular paths for approaching and departing the paths depend on **Q220 CORNER RADIUS**.
- The control calculates the offset from the programmed width, the tool radius, the maximum path overlap factor **Q370** and the set-up clearance to the side **Q357**.

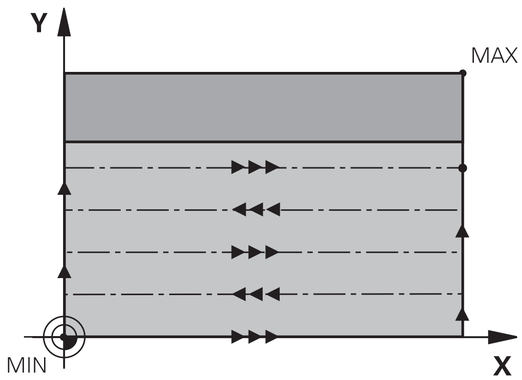
Strategy Q389=4



Cycle sequence

1 From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane. The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.
2 The control then positions the tool at rapid traverse **FMAX** to set-up clearance in the spindle axis.
3 The tool then moves in the spindle axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control.
4 The tool subsequently moves to the starting point of the milling path at the programmed **Feed rate for milling** on a tangential approach path.
5 The control machines the level surface at the feed rate for milling from the outside toward the inside with ever-shorter milling paths. The constant stepover results in the tool being continuously engaged.
6 The process is repeated until the programmed surface has been machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**.
7 If more than one infeed is required, the control moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.
8 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**.

Limits



The limits enable you to set limits to the machining of the level surface so that, for example, side walls or shoulders are considered during machining. A side wall that is defined by a limit is machined to the finished dimension resulting from the starting point or the side lengths of the level surface. During roughing the control takes the allowance for the side into account, whereas during finishing the allowance is used for pre-positioning the tool.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface!

- Enter depth as negative
- Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- Cycle **233** monitors the entries made for the tool or cutting edge length in **LCUTS** in the tool table. If the tool or cutting edge length is not sufficient for a finishing operation, the control will subdivide the process into multiple machining steps.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.

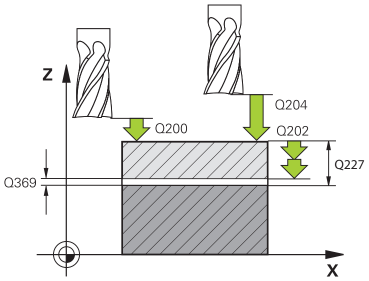
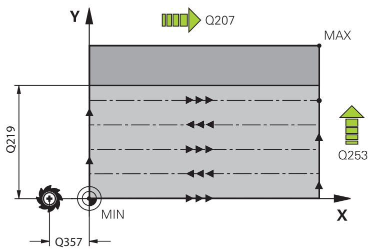
Notes on programming

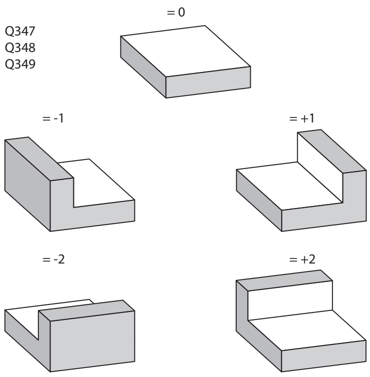
- Pre-position the tool in the working plane to the starting position with radius compensation R0. Note the machining direction.
- If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).
- If you define **Q370 TOOL PATH OVERLAP** >1, the programmed overlap factor will be taken into account right from the first machining path.
- If a limit (**Q347**, **Q348** or **Q349**) was programmed in the machining direction **Q350**, the cycle will extend the contour in the infeed direction by corner radius **Q220**. The specified surface will be machined completely.

> Enter **Q204 2ND SET-UP CLEARANCE** in such a way that no collision with the workpiece or the fixtures can occur.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	Q389 Machining strategy (0-4)? Specify how the control machines the surface: **0**: Meander machining, stepover at positioning feed rate outside the surface to be machined **1**: Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined **2**: Machining line by line, retraction and stepover at positioning feed rate outside the surface to be machined **3**: Machining line by line, retraction and stepover at positioning feed rate at the edge of the surface to be machined **4**: Helical machining, uniform infeed from the outside toward the inside Input: **0**, **1**, **2**, **3**, **4**
	Q350 Milling direction? Axis in the working plane that defines the machining direction: **1**: Main axis = Machining direction **2**: Secondary axis = Machining direction Input: **1**, **2**
	Q218 First side length? Length of the surface to be machined in the main axis of the working plane, referencing the starting point in the 1st axis. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q219 Second side length? Length of the surface to be machined in the secondary axis of the working plane. Use algebraic signs to specify the direction of the first cross feed referenced to the **STARTNG PNT 2ND AXIS**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	**Q227 Starting point in 3rd axis?** Coordinate of the workpiece surface used to calculate the infeeds. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q386 End point in 3rd axis? Coordinate in the spindle axis on which the surface will be face-milled. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q369 Finishing allowance for floor? Value used for the last infeed. This value has an incremental effect. Input: **0...99999.9999**
	Q202 Maximum plunging depth? Infeed per cut. Enter an incremental value greater than 0. Input: **0...99999.9999**
	Q370 Path overlap factor? Maximum stepover factor k. The control calculates the actual stepover from the second side length (**Q219**) and the tool radius so that a constant stepover is used for machining. Input: **0.0001...1.9999**
	**Q207 Feed rate for milling?** Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min while milling the last infeed Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely inside the material (**Q389**=1), the control uses the cross feed rate for milling **Q207**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q357 Safety clearance to the side? Parameter **Q357** influences the following situations: **Approaching the first infeed depth: Q357** is the lateral distance from the tool to the workpiece. **Roughing with the Q389 = 0 to 3 roughing strategies:** The surface to be machined is extended in **Q350 MILLING DIRECTION** by the value from **Q357** if no limit has been set in that direction. **Side finishing:** The paths are extended by **Q357** in the **Q350 MILLING DIRECTION**. This value has an incremental effect. Input: **0...99999.9999**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	**Q347 1st limit?** Select the side of the workpiece where the plane surface is bordered by a side wall (not possible with helical machining). Depending on the position of the side wall, the control limits the machining of the plane surface to the corresponding starting point coordinate or side length: **0**: No limitation **-1**: Limit in negative main axis **+1**: Limit in positive main axis **-2**: Limit in negative secondary axis **+2**: Limit in positive secondary axis Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q348 2nd limit? See parameter **Q347** 1st limit Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q349 3rd limit? See parameter **Q347** 1st limit Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q220 Corner radius? Radius of a corner at limits (**Q347** to **Q349**) Input: **0...99999.9999**

Help graphic	Parameter
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	Q367 Surface position (-1/0/1/2/3/4)? Position of the surface relative to the position of the tool when the cycle is called: **-1**: Tool position = Current position **0**: Tool position = Center of stud **1**: Tool position = Lower left corner **2**: Tool position = Lower right corner **3**: Tool position = Upper right corner **4**: Tool position = Upper left corner Input: **-1**, **0**, **+1**, **+2**, **+3**, **+4**

Example

```
11 CYCL DEF 233 FACE MILLING ~
    Q215=+0                     ;MACHINING OPERATION ~
    Q389=+2                     ;MILLING STRATEGY ~
    Q350=+1                     ;MILLING DIRECTION ~
    Q218=+60                    ;FIRST SIDE LENGTH ~
    Q219=+20                    ;2ND SIDE LENGTH ~
    Q227=+0                     ;STARTNG PNT 3RD AXIS ~
    Q386=+0                     ;END POINT 3RD AXIS ~
    Q369=+0                     ;ALLOWANCE FOR FLOOR ~
    Q202=+5                     ;MAX. PLUNGING DEPTH ~
    Q370=+1                     ;TOOL PATH OVERLAP ~
    Q207=+500                   ;FEED RATE MILLING ~
    Q385=+500                   ;FINISHING FEED RATE ~
    Q253=+750                   ;F PRE-POSITIONING ~
    Q357=+2                     ;CLEARANCE TO SIDE ~
    Q200=+2                     ;SET-UP CLEARANCE ~
    Q204=+50                    ;2ND SET-UP CLEARANCE ~
    Q347=+0                     ;1ST LIMIT ~
    Q348=+0                     ;2ND LIMIT ~
    Q349=+0                     ;3RD LIMIT ~
    Q220=+0                     ;CORNER RADIUS ~
    Q368=+0                     ;ALLOWANCE FOR SIDE ~
    Q338=+0                     ;INFEED FOR FINISHING ~
    Q367=-1                     ;SURFACE POSITION
12 L  X+50  Y+50 R0 FMAX M99
```

14.3.24 SL cycles

General information

SL Cycles enable you to form complex contours by combining up to twelve subcontours (pockets or islands). You define the individual subcontours in subprograms. The control calculates the entire contour from the list of subcontours (subprogram numbers) you have specified in Cycle **14 CONTOUR**.

Programming and operating notes:

- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- SL Cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always use the simulation to verify your program before running it. This is a simple way of finding out whether the program calculated by the control will provide the desired results.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Characteristics of the subprograms

- Closed contour without approach and departure movements
- Coordinate transformations are permitted; if they are programmed within the subcontours, they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The control recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR
- The control recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL
- The subprograms must not contain spindle axis coordinates.
- Always program both axes in the first NC block of the subprogram
- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms
- Without machining cycles, feed rates, and M functions

Cycle properties

- The control automatically positions the tool to the set-up clearance before each cycle. You must move the tool to a safe position before the cycle call
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them
- The radius of inside corners can be programmed—the tool will not stop, dwell marks are avoided (this applies to the outermost path of roughing or side finishing operations)
- The contour is approached on a tangential arc for side finishing
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for spindle axis Z, for example, the arc is in the Z/X plane)
- The contour is machined throughout in either climb or up-cut milling

The machining data, such as milling depth, allowances, and set-up clearance can be entered centrally in Cycle **20 CONTOUR DATA**.

Program structure: Machining with SL Cycles

```
0 BEGIN SL 2 MM
...
12 CYCL DEF 14 CONTOUR
...
13 CYCL DEF 20 CONTOUR DATA
...
16 CYCL DEF 21 PILOT DRILLING
...
17 CYCL CALL
...
22 CYCL DEF 23 FLOOR FINISHING
...
23 CYCL CALL
...
26 CYCL DEF 24 SIDE FINISHING
...
27 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM SL2 MM
```

14.3.25 Cycle 20 CONTOUR DATA

Application

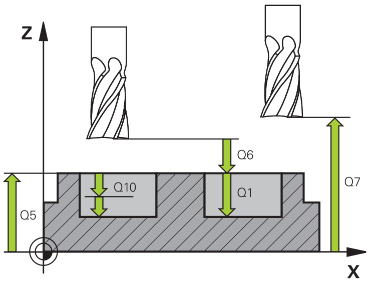
Use Cycle **20** to specify machining data for the subprograms describing the subcontours.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **20** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **20** are valid for Cycles **21** to **24**.
- If you are using the SL cycles in **Q** parameter programs, the cycle parameters **Q1** to **Q20** cannot be used as program parameters.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH = 0, the control performs the cycle at the depth 0.

Cycle parameters

Help graphic | **Parameter**



Q1 Milling depth?

Distance between workpiece surface and pocket floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q2 Path overlap factor?

Q2 x tool radius = stepover factor k

Input: **0.0001...1.9999**

Q3 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q4 Finishing allowance for floor?

Finishing allowance for the floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q5 Workpiece surface coordinate?

Absolute coordinate of the top surface of the workpiece

Input: **-99999.9999...+99999.9999**

Q6 Set-up clearance?

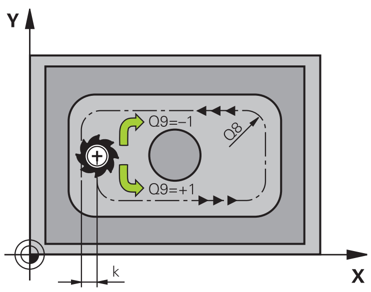
Distance between tool tip and the top surface of the workpiece. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q7 Clearance height?

Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). The value has an absolute effect.

Input: **-99999.9999...+99999.9999**



Q8 Inside corner radius?:

Inside "corner" rounding radius; entered value is referenced to the path of the tool center and is used to calculate smoother traverse motions between the contour elements.

Q8 is not a radius that is inserted between programmed elements as a separate contour element.

Input: **0...99999.9999**

Q9 Direction of rotation? cw = -1

Machining direction for pockets

Q9 = –1 up-cut milling for pocket and island

Q9 = +1 climb milling for pocket and island

Input: **-1**, **0**, **+1**

Example

11 CYCL DEF 20 CONTOUR DATA ~	
Q1=-20	;MILLING DEPTH ~
Q2=+1	;TOOL PATH OVERLAP ~
Q3=+0.2	;ALLOWANCE FOR SIDE ~
Q4=+0.1	;ALLOWANCE FOR FLOOR ~
Q5=+0	;SURFACE COORDINATE ~
Q6=+2	;SET-UP CLEARANCE ~
Q7=+50	;CLEARANCE HEIGHT ~
Q8=+0	;ROUNDING RADIUS ~
Q9=+1	;ROTATIONAL DIRECTION

14.3.26 Cycle 21 PILOT DRILLING

Application

Use Cycle **21 PILOT DRILLING** if you machine a contour and then use a tool for roughing it out which has no center-cut end mill (ISO 1641). This cycle drills a hole in the area that will be roughed out later with a cycle such as Cycle **22**. Cycle **21** takes the finishing allowance for side and the finishing allowance for floor as well as the radius of the rough-out tool into account for the cutter infeed points. The cutter infeed points also serve as starting points for roughing.

Before programming the call of Cycle **21** you need to program two further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**—required by Cycle **21 PILOT DRILLING** to determine the drilling position in the plane
- Cycle **20 CONTOUR DATA**—required by Cycle **21 PILOT DRILLING** to determine parameters such as the hole depth and the set-up clearance

Cycle sequence

1 The control first positions the tool in the plane (the position results from the contour that you previously defined with Cycle **14** or **SEL CONTOUR**, and from the information on the rough-out tool)
2 The tool then moves at rapid traverse **FMAX** to set-up clearance. (specify the set-up clearance in Cycle **20 CONTOUR DATA**)
3 The tool drills from the current position to the first plunging depth at the programmed feed rate **F**.
4 Then, the tool retracts at rapid traverse **FMAX** to the starting position and advances again to the first plunging depth minus the advanced stop distance t
5 The advanced stop distance is automatically calculated by the control:
   - At a total hole depth up to 30 mm: t = 0.6 mm
   - At a total hole depth exceeding 30 mm: t = hole depth / 50
   - Maximum advanced stop distance: 7 mm
6 The tool then advances with another infeed at the programmed feed rate **F**.
7 The control repeats this procedure (steps 1 to 4) until the total hole depth is reached. The finishing allowance for floor is taken into account
8 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

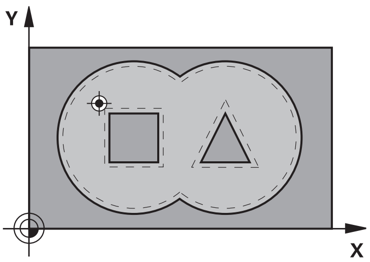
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- When calculating the infeed points, the control does not account for the delta value **DR** programmed in a **TOOL CALL** block.
- In narrow areas, the control may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.
- If **Q13**=0, the control uses the data of the tool that is currently in the spindle.

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining. After the end of the cycle, do not position the tool in the plane incrementally, but rather to an absolute position if you have programmed **ToolAxClearanceHeight**.

Cycle parameters

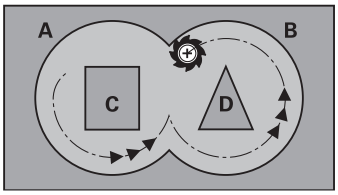
Help graphic	Parameter
	**Q10 Plunging depth?** Tool infeed per cut (minus sign for negative machining direction). This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Tool traversing speed in mm/min during plunging Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q13 or **QS13 Rough-out tool number/name?** Number or name of the rough-out tool. You are able to transfer the tool directly from the tool table via the selection option in the action bar. Input: **0...999999.9** or max. **255** characters

Example

```
11 CYCL DEF 21 PILOT DRILLING ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q13=+0                  ;ROUGH-OUT TOOL
```

14.3.27 Cycle 22 ROUGH-OUT

Application



Use Cycle **22 ROUGH-OUT** to define the technology data for roughing.

Before programming the call of Cycle **22**, you need to program further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **21 PILOT DRILLING**, if applicable

Cycle sequence

1. The control positions the tool above the cutter infeed point, taking the finishing allowance for side into account
2. After reaching the first plunging depth, the tool mills the contour in an outward direction at the programmed milling feed rate **Q12**
3. The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B)
4. In the next step, the control moves the tool to the next plunging depth and repeats the roughing procedure until the program depth is reached
5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- During fine roughing, the control does not take a defined wear value **DR** of the coarse roughing tool into account.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q1**, the control will display an error message.

This cycle might require a center-cut end mill (ISO 1641) or pilot drilling with Cycle **21**.

Notes on programming

- If you clear out an acute inside corner and use an overlap factor greater than 1, some material might be left over. Check especially the innermost path in the test run graphic and, if necessary, change the overlap factor slightly. This allows another distribution of cuts, which often provides the desired results.
- Define the plunging behavior of Cycle **22** with parameter **Q19** and in the **ANGLE** and **LCUTS** columns of the tool table:
  - If **Q19** = 0 is defined, the tool will always plunge perpendicularly, even if a plunge angle (**ANGLE**) was defined for the active tool
  - If you define **ANGLE** = 90°, the control will plunge perpendicularly. The reciprocation feed rate **Q19** is used as plunging feed rate
  - If the reciprocation feed rate **Q19** is defined in Cycle **22** and **ANGLE** is between 0.1 and 89.999 in the tool table, the tool plunges helically using the defined **ANGLE**
  - If the reciprocation feed is defined in Cycle **22** and no **ANGLE** is defined in the tool table, the control displays an error message
  - If the geometry conditions do not allow helical plunging (slot geometry), the control tries a reciprocating plunge (the reciprocation length is calculated from **LCUTS** and **ANGLE** (reciprocation length = **LCUTS** / tan **ANGLE**))

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket.
  - **PosBeforeMachining**: Return to starting position
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

Cycle parameters

Help graphic	Parameter
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q18 or **QS18 Coarse roughing tool?** Number or name of the tool with which the control has already coarse-roughed the contour. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE**. Input: **0...99999.9** or max. **255** characters
	Q19 Feed rate for reciprocation? Reciprocation feed rate in mm/min Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q208 Feed rate for retraction? Tool traversing speed in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q12**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q401 Feed rate factor in %? Percentage value to which the control reduces the machining feed rate (**Q12**) as soon as the tool moves with its entire circumference within the material during roughing. If you use the feed rate reduction, then you can define the feed rate for roughing so large that there are optimum cutting conditions with the path overlap (**Q2**) specified in Cycle **20**. The control then reduces the feed rate as per your definition at transitions and narrow places, reducing the total machining time. Input: **0.0001...100**
	Q404 Fine roughing strategy (0/1)? Define how the control will move the tool during fine roughing when the radius of the fine-roughing tool is equal to or larger than half the radius of the coarse roughing tool. **0**: Between areas that need to be fine-roughed, the control moves the tool along the contour at the current depth **1**: Between areas that need to be fine-roughed, the control retracts the tool to set-up clearance and then moves it to the starting point of the next area to be roughed out Input: **0**, **1**

Example

```
11 CYCL DEF 22 ROUGH-OUT ~
    Q10=-5              ;PLUNGING DEPTH ~
    Q11=+150            ;FEED RATE FOR PLNGNG ~
    Q12=+500            ;FEED RATE F. ROUGHNG ~
    Q18=+0              ;COARSE ROUGHING TOOL ~
    Q19=+0              ;FEED RATE FOR RECIP. ~
    Q208=+99999         ;RETRACTION FEED RATE ~
    Q401=+100           ;FEED RATE FACTOR ~
    Q404=+0             ;FINE ROUGH STRATEGY
```

14.3.28 Cycle 23 FLOOR FINISHING

Application

With Cycle **23 FLOOR FINISHING**, you can finish your contour by taking the finishing allowance for the floor into account that has been programmed in Cycle **20**. The tool smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control moves the tool to depth vertically. The tool then clears the finishing allowance remaining from rough-out.

Before programming the call of Cycle **23**, you need to program further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **21 PILOT DRILLING**, if applicable
- Cycle **22 ROUGH-OUT**, if necessary

Cycle sequence

1. The control positions the tool to the clearance height at rapid traverse FMAX.
2. The tool then moves in the tool axis at the feed rate **Q11**.
3. The tool smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control moves the tool to depth vertically
4. The tool clears the finishing allowance remaining from rough-out.
5. Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

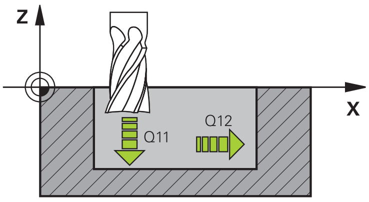
- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.
- The approaching radius for pre-positioning to the final depth is permanently defined and independent of the plunging angle of the tool.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q15**, the control will display an error message.

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket.
  - **PosBeforeMachining**: Return to starting position
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

Cycle parameters

Help graphic	Parameter
	**Q11 Feed rate for plunging?** Tool traversing speed in mm/min during plunging Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q208 Feed rate for retraction? Tool traversing speed in mm/min when retracting after the machining operation. If you enter **Q208** = 0, the control retracts the tool at the feed rate specified in **Q12**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Example

```
11 CYCL DEF 23 FLOOR FINISHING ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q12=+500                ;FEED RATE F. ROUGHNG ~
    Q208=+99999             ;RETRACTION FEED RATE
```

14.3.29 Cycle 24 SIDE FINISHING

Application

Cycle **24 SIDE FINISHING** allows you to finish your contour by taking the side finishing allowance into account that has been programmed in Cycle **20**. You can run this cycle in climb or up-cut milling mode.

Before programming the call of Cycle **24**, you need to program further cycles:

- Cycle **14 CONTOUR** or **SEL CONTOUR**
- Cycle **20 CONTOUR DATA**
- Cycle **21 PILOT DRILLING**, if applicable
- Cycle **22 ROUGH-OUT**, if necessary

Cycle sequence

1 The control positions the tool above the workpiece surface to the starting point for the approach position. This position in the plane results from a tangential arc on which the control moves the tool when approaching the contour
2 The control then moves the tool to the first plunging depth using the feed rate for plunging
3 The contour is approached on a tangential arc and machined up to the end. Each subcontour is finished separately
4 The tool moves on a tangential helical arc when approaching the finishing contour or retracting from it. The starting height of the helix is 1/25 of the set-up clearance **Q6**, but max. the remaining last plunging depth above the final depth
5 Finally, the tool retracts in the tool axis to the clearance height or to the position last programmed before the cycle. This behavior depends on the machine parameter **posAfterContPocket** (no. 201007).

i	The starting point calculated by the control also depends on the machining sequence. If you select the finishing cycle with the **GOTO** key and then start the NC program, the starting point can be at a different location from where it would be if you execute the NC program in the defined sequence.

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If no allowance was defined in Cycle **20**, the control generates the error message "Tool radius too large."
- If you run Cycle **24** without having roughed out with Cycle **22**, then enter "0" for the radius of the rough mill.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket and the allowance programmed in Cycle **20**.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q15**, the control will display an error message.
- You can execute this cycle using a grinding tool.

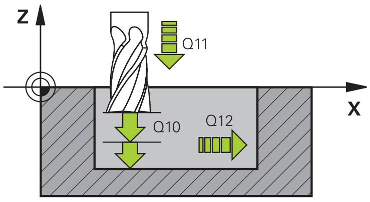
Notes on programming

- The sum of finishing allowance for the side (**Q14**) and the radius of the finish mill must be smaller than the sum of allowance for side (**Q3**, Cycle **20**) and the radius of the rough mill.
- The finishing allowance for the side **Q14** is left over after finishing. Therefore, it must be smaller than the allowance in Cycle **20**.
- Cycle **24** can also be used for contour milling. In that case, you must do the following:
  - Define the contour to be milled as a single island (without pocket boundary)
  - In Cycle **20**, enter a finishing allowance (**Q3**) greater than the sum of the finishing allowance **Q14** + radius of the tool being used

Note regarding machine parameters

- Use the machine parameter **posAfterContPocket** (no. 201007) to define how to move the tool after machining the contour pocket:
  - **PosBeforeMachining**: Return to starting position.
  - **ToolAxClearanceHeight**: Position the tool axis to clearance height.

Cycle parameters

Help graphic	Parameter
	Q9 Direction of rotation? cw = -1 Machining direction: **+1**: Counterclockwise **–1**: Clockwise Input: **-1**, **+1**
 	**Q10 Plunging depth?** Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Tool traversing speed in mm/min during plunging Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q14 Finishing allowance for side? The finishing allowance for the side **Q14** is left over after finishing. This allowance must be smaller than the allowance in Cycle **20**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **Q438 = –1**: The control assumes that the tool last used is the rough-out tool (default behavior) **Q438 = 0**: If there was no coarse-roughing, enter the number of a tool with the radius 0. This is usually the tool numbered 0. Input: **-1...+32767.9** or **255** characters

Example

11 CYCL DEF 24 SIDE FINISHING ~	
Q9=+1	;ROTATIONAL DIRECTION ~
Q10=+5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q12=+500	;FEED RATE F. ROUGHNG ~
Q14=+0	;ALLOWANCE FOR SIDE ~
Q438=-1	;ROUGH-OUT TOOL

14.3.30 Cycle 270 CONTOUR TRAIN DATA

Application

You can use this cycle to specify various properties of Cycle **25 CONTOUR TRAIN**.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **270** is DEF-active, which means that it becomes effective as soon as it is defined in the NC program.
- If Cycle **270** is used, do not define any radius compensation in the contour subprogram.
- Define Cycle **270** before Cycle **25**.

Cycle parameters

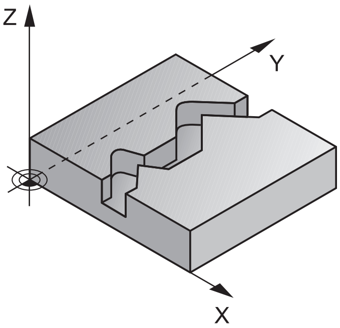
Help graphic	Parameters
	Q390 Type of approach/departure? Definition of type of approach/departure: **1**: Approach the contour tangentially on a circular arc **2**: Approach the contour tangentially on a straight line **3**: Approach the contour at a right angle **0** and **4**: No approach or departure movement is performed. Input: **1**, **2**, **3**
	Q391 Radius comp. (0=R0/1=RL/2=RR)? Definition of radius compensation: **0**: Machine the defined contour without radius compensation **1**: Machine the defined contour with compensation to the left **2**: Machine the defined contour with compensation to the right Input: **0**, **1**, **2**
	Q392 App. radius/dep. radius? Only in effect if a tangential approach on a circular path was selected (**Q390** = 1). Radius of the approach/departure arc Input: **0...99999.9999**
	Q393 Center angle? Only in effect if a tangential approach on a circular path was selected (**Q390** = 1). Angular length of the approach arc Input: **0...99999.9999**
	Q394 Distance from aux. point? Only in effect if a tangential approach on a straight line or a right-angle approach is selected (**Q390** = 2 or **Q390** = 3). Distance to the auxiliary point from which the tool will approach the contour. Input: **0...99999.9999**

Example

```
11 CYCL DEF 270 CONTOUR TRAIN DATA ~
    Q390=+1                 ;TYPE OF APPROACH ~
    Q391=+1                 ;RADIUS COMPENSATION ~
    Q392=+5                 ;RADIUS ~
    Q393=+90                ;CENTER ANGLE ~
    Q394=+0                 ;DISTANCE
```

14.3.31 Cycle 25 CONTOUR TRAIN

Application



In conjunction with Cycle **14 CONTOUR**, this cycle enables you to machine open and closed contours.

Cycle **25 CONTOUR TRAIN** offers considerable advantages over machining a contour using positioning blocks:

- The control monitors the operation to prevent undercuts and contour damage (run a graphic simulation of the contour before execution)
- If the radius of the selected tool is too large, the corners of the contour may have to be reworked
- Machining can be done throughout by up-cut or by climb milling. The type of milling will even be retained if the contours were mirrored
- The tool can traverse back and forth for milling in several infeeds: This results in faster machining
- Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control takes only the first label of Cycle **14 CONTOUR** into account.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.
- You can execute this cycle using a grinding tool.

Notes on programming

- Cycle **20 CONTOUR DATA**, is not required.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q1 Milling depth? Distance between workpiece surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q5 Workpiece surface coordinate? Absolute coordinate of the top surface of the workpiece Input: **-99999.9999...+99999.9999**
	Q7 Clearance height? Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q15 Climb or up-cut? up-cut = -1 **+1**: Climb milling **-1**: Up-cut milling **0**: Climb milling and up-cut milling alternately in several infeeds Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q18 or **QS18 Coarse roughing tool?** Number or name of the tool with which the control has already coarse-roughed the contour. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE**. Input: **0...99999.9** or max. **255** characters
	Q446 Accepted residual material? Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input: **0.001...9.999**
	Q447 Maximum connection distance? Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input: **0...999.999**
	Q448 Path extension? Length by which the tool path is extended at the beginning and end of a contour area. The control always extends the tool path in parallel to the contour. Input: **0...99.999**

Example

```
11 CYCL DEF 25 CONTOUR TRAIN ~
    Q1=-20                  ;MILLING DEPTH ~
    Q3=+0                   ;ALLOWANCE FOR SIDE ~
    Q5=+0                   ;SURFACE COORDINATE ~
    Q7=+50                  ;CLEARANCE HEIGHT ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q12=+500                ;FEED RATE F. ROUGHNG ~
    Q15=+1                  ;CLIMB OR UP-CUT ~
    Q18=+0                  ;COARSE ROUGHING TOOL ~
    Q446=+0.01              ;RESIDUAL MATERIAL ~
    Q447=+10                ;CONNECTION DISTANCE ~
    Q448=+2                 ;PATH EXTENSION
```

14.3.32 Cycle 275 TROCHOIDAL SLOT

Application

In conjunction with Cycle **14 KONTUR**, this cycle enables you to completely machine open and closed slots or contour slots using trochoidal milling.

With trochoidal milling, large cutting depths and high cutting speeds can be combined as the equally distributed cutting forces prevent increased wear of the tool. When indexable inserts are used, the entire cutting length is exploited to increase the attainable chip volume per tooth. Moreover, trochoidal milling is easy on the machine mechanics. Enormous amounts of time can also be saved by combining this milling method with the integrated adaptive feed control **AFC** (option 45).

Further information: "Adaptive Feed Control (AFC, option 45)", Page 1114

Depending on the cycle parameters you select, the following machining alternatives are available:

- Complete machining: Roughing, side finishing
- Only roughing
- Only side finishing

Program structure: Machining with SL Cycles

```
0 BEGIN CYC275 MM
...
12 CYCL DEF 14 CONTOUR
...
13 CYCL DEF 275 TROCHOIDAL SLOT
...
14 CYCL CALL M3
...
50 L Z+250 R0 FMAX M2
51 LBL 10
...
55 LBL 0
...
99 END PGM CYC275 MM
```

Cycle sequence

Roughing closed slots

In case of a closed slot, the contour description must always start with a straight-line block (**L** block).

1 Following the positioning logic, the tool moves to the starting point of the contour description and moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. Specify the plunging strategy with parameter **Q366**.
2 The control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in the machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached

Finishing closed slots

5 If a finishing allowance has been defined, the control finishes the slot walls, in multiple infeeds, if so specified. Starting from the defined starting point, the control approaches the slot wall tangentially. Climb or up-cut milling is taken into consideration.

Roughing open slots

The contour description of an open slot must always start with an approach block (**APPR**).

1 Following the positioning logic, the tool moves to the starting point of the machining operation as defined by the parameters in the **APPR** block and plunges vertically to the first plunging depth.
2 The control roughs the slot in circular motions until the contour end point is reached. During the circular motion, the control moves the tool in the machining direction by an infeed you can define (**Q436**). Define climb or up-cut of the circular motion in parameter **Q351**.
3 At the contour end point, the control moves the tool to clearance height and returns it to the starting point of the contour description.
4 This process is repeated until the programmed slot depth is reached

Finishing open slots

5 If a finishing allowance has been defined, the control finishes the slot walls (in multiple infeeds if specified). The control approaches the slot wall starting from the defined starting point of the **APPR** block. Climb or up-cut milling is taken into consideration

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

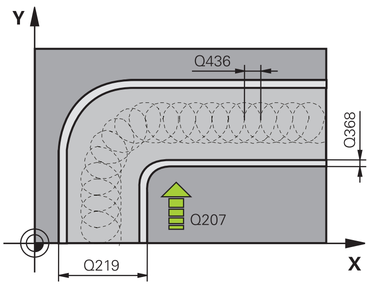
- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

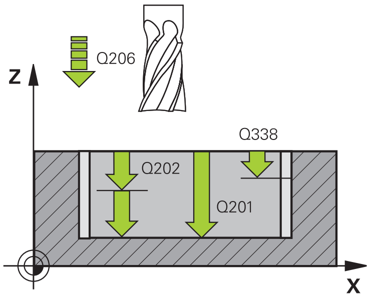
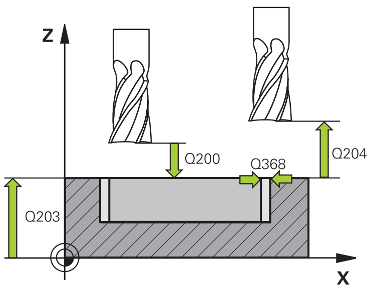
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- In conjunction with Cycle **275**, the control does not require Cycle **20 CONTOUR DATA**.

Notes on programming

- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- If using Cycle **275 TROCHOIDAL SLOT**, you may define only one contour subprogram in Cycle **14 CONTOUR**.
- Define the center line of the slot with all available path functions in the contour subprogram.
- The starting point of a closed slot must not be located in a contour corner.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing Side finishing and floor finishing are only executed if the respective finishing allowance (**Q368**, **Q369**) has been defined Input: **0**, **1**, **2**
	**Q219 Width of slot?** Enter the width of the slot. It is in parallel to the secondary axis of the working plane. If you enter a slot width that equals the tool diameter, then the control will carry out the roughing process only (slot milling). Maximum slot width for roughing: Twice the tool diameter Input: **0...99999.9999**
	Q368 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q436 Feed per revolution? Value by which the control moves the tool in the machining direction per revolution. The value has an absolute effect. Input: **0...99999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**

Help graphic	Parameter
	**Q201 Depth?** Distance between workpiece surface and slot floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q202 Plunging depth? Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect. Input: **0...99999.9999**
	Q206 Feed rate for plunging? Traversing speed of the tool in mm/min for moving to depth Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q338 Infeed for finishing? Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side and floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	**Q200 Set-up clearance?** Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0** = Vertical plunging. The control plunges perpendicularly, regardless of the plunging angle ANGLE defined in the tool table **1** = No function **2**= Reciprocating plunge. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message Input: **0**, **1**, **2** or **PREDEF**

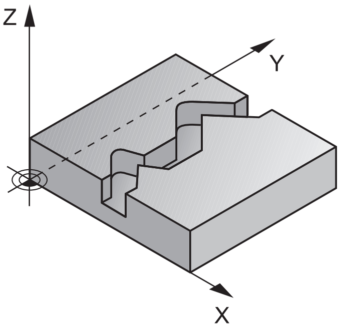
Help graphic	Parameter
	Q369 Finishing allowance for floor? Finishing allowance for the floor. This value has an incremental effect. Input: **0...99999.9999**
	Q439 Feed rate reference (0-3)? Specify the reference for the programmed feed rate: **0**: Feed rate is referenced to the path of the tool center **1**: Feed rate is referenced to the cutting edge only during side finishing; otherwise, it is referenced to the path of the tool center **2**: Feed rate is referenced to the cutting edge during side finishing **and** floor finishing; otherwise it is referenced to the path of the tool center **3**: Feed rate is always referenced to the cutting edge Input: **0**, **1**, **2**, **3**

Example

```
11 CYCL DEF 275 TROCHOIDAL SLOT ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q219=+10                ;SLOT WIDTH ~
    Q368=+0                 ;ALLOWANCE FOR SIDE ~
    Q436=+2                 ;INFEED PER REV. ~
    Q207=+500               ;FEED RATE MILLING ~
    Q351=+1                 ;CLIMB OR UP-CUT ~
    Q201=-20                ;DEPTH ~
    Q202=+5                 ;PLUNGING DEPTH ~
    Q206=+150               ;FEED RATE FOR PLNGNG ~
    Q338=+0                 ;INFEED FOR FINISHING ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q204=+50                ;2ND SET-UP CLEARANCE ~
    Q366=+2                 ;PLUNGE ~
    Q369=+0                 ;ALLOWANCE FOR FLOOR ~
    Q439=+0                 ;FEED RATE REFERENCE
12 CYCL CALL
```

14.3.33 Cycle 276 THREE-D CONT. TRAIN

Application



In conjunction with Cycle **14 CONTOUR** and Cycle **270 CONTOUR TRAIN DATA**, this cycle enables you to machine open and closed contours. You can also work with automatic residual material detection. This way you can subsequently complete e.g. inside corners with a smaller tool.

In contrast to Cycle **25 CONTOUR TRAIN**, Cycle **276 THREE-D CONT. TRAIN** also processes tool axis coordinates defined in the contour subprogram. This cycle can thus machine three-dimensional contours.

We recommend that you program Cycle **270 CONTOUR TRAIN DATA** before Cycle **276 THREE-D CONT. TRAIN**.

Cycle sequence

Machining a contour without infeed: Milling depth Q1 = 0

1. The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle **270 CONTOUR TRAIN DATA** (e.g., the Type of approach). The control then moves the tool to the first plunging depth
2. According to the previously defined Cycle **270 CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3. At the end of the contour, the tool will be retracted as defined in Cycle **270 CONTOUR TRAIN DATA**
4. Finally, the control retracts the tool to the clearance height.

Machining a contour with infeed: Milling depth Q1 not equal to 0 and plunging depth Q10 are defined

1. The tool traverses to the starting point of machining. This starting point results from the first contour point, the selected milling mode (climb or up-cut) and the parameters from the previously defined Cycle **270 CONTOUR TRAIN DATA** (e.g., the Type of approach). The control then moves the tool to the first plunging depth
2. According to the previously defined Cycle **270 CONTOUR TRAIN DATA**, the tool approaches the contour and then machines it completely to the end
3. If you selected machining with climb milling and up-cut milling (**Q15** = 0), the control will perform a reciprocation movement. The infeed movement (plunging) will be performed at the end and at the starting point of the contour. If **Q15** is not equal to 0, the tool is moved to clearance height and is returned to the starting point of machining. From there, the control moves the tool to the next plunging depth
4. The departure will be performed as defined in Cycle **270 CONTOUR TRAIN DATA**
5. This process is repeated until the programmed depth is reached.
6. Finally, the control retracts the tool to the clearance height

Notes

NOTICE

Danger of collision!

If you have set the **posAfterContPocket** parameter (no. 201007) to **ToolAxClearanceHeight**, the control will position the tool at clearance height only in the direction of the tool axis when the cycle has finished. The control will not position the tool in the working plane.

- After the end of the cycle, position the tool with all coordinates of the working plane, e.g. **L X+80 Y+0 R0 FMAX**
- Make sure to program an absolute position after the cycle; do not program an incremental traversing movement

NOTICE

Danger of collision!

A collision may occur if you position the tool behind an obstacle before the cycle is called.

- Before the cycle call, position the tool in such a way that the tool can approach the starting point of the contour without collision
- If the position of the tool is below the clearance height when the cycle is called, the control will issue an error message

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program **APPR** and **DEP** blocks for contour approach and departure, the control monitors whether the execution of any of these blocks would damage the contour.
- If using Cycle **25 CONTOUR TRAIN**, you can define only one subprogram in Cycle **14 CONTOUR**.
- We recommend that you use Cycle **270 CONTOUR TRAIN DATA** in conjunction with Cycle **276**. Cycle **20 CONTOUR DATA**, however, is not required.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- If **M110** is activated during operation, the feed rate for arcs compensated on the inside will be reduced accordingly.

Notes on programming

- The first NC block in the contour subprogram must contain values in all of the three axes X, Y and Z.
- The algebraic sign for the depth parameter determines the working direction. If you program DEPTH = 0, the control will use the tool axis coordinates that have been specified in the contour subprogram.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q1 Milling depth? Distance between workpiece surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the working plane. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q7 Clearance height? Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q15 Climb or up-cut? up-cut = -1 **+1**: Climb milling **-1**: Up-cut milling **0**: Climb milling and up-cut milling alternately in several infeeds Input: **-1**, **0**, **+1**
	Q18 or **QS18 Coarse roughing tool?** Number or name of the tool with which the control has already coarse-roughed the contour. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. If there was no coarse roughing, enter "0"; if you enter a number or a name, the control will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion to be roughed cannot be approached from the side, the control will mill in a reciprocating plunge-cut; for this purpose you must enter the tool length **LCUTS** in the TOOL.T tool table and define the maximum plunging angle of the tool with **ANGLE**. Input: **0...99999.9** or max. **255** characters

Help graphic	Parameter
	Q446 Accepted residual material? Specify the maximum value in mm up to which you accept residual material on the contour. For example, if you enter 0.01 mm, the control will stop machining residual material when it has reached a thickness of 0.01 mm. Input: **0.001...9.999**
	Q447 Maximum connection distance? Maximum distance between two areas to be fine-roughed. Within this distance, the tool will move along the contour without lift-off movement, remaining at machining depth. Input: **0...999.999**
	Q448 Path extension? Length by which the tool path is extended at the beginning and end of a contour area. The control always extends the tool path in parallel to the contour. Input: **0...99.999**

Example

```
11 CYCL DEF 276 THREE-D CONT. TRAIN ~
    Q1=-20              ;MILLING DEPTH ~
    Q3=+0               ;ALLOWANCE FOR SIDE ~
    Q7=+50              ;CLEARANCE HEIGHT ~
    Q10=-5              ;PLUNGING DEPTH ~
    Q11=+150            ;FEED RATE FOR PLNGNG ~
    Q12=+500            ;FEED RATE F. ROUGHNG ~
    Q15=+1              ;CLIMB OR UP-CUT ~
    Q18=+0              ;COARSE ROUGHING TOOL ~
    Q446=+0.01          ;RESIDUAL MATERIAL ~
    Q447=+10            ;CONNECTION DISTANCE ~
    Q448=+2             ;PATH EXTENSION
```

14.3.34 OCM cycles

OCM cycles

General information

Refer to your machine manual.
Your machine manufacturer enables this function.

Using OCM cycles (**Optimized Contour Milling**), you can combine subcontours to form complex contours. These cycles provide more functionality than Cycles **22** to **24**. The OCM cycles feature the following additional functions:

- When roughing, the control will maintain the specified tool angle precisely
- Besides pockets, you can also machine islands and open pockets

Programming and operating notes:

- You can program up to 16 384 contour elements in one OCM cycle.
- OCM cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always verify the program graphically! This is a simple way of finding out whether the program calculated by the control will provide the desired results.

Contact angle

When roughing, the control will retain the tool angle precisely. The tool angle can be defined implicitly by specifying an overlap factor. The maximum overlap factor is 1.99; this corresponds to an angle of nearly 180°.

Contour

Specify the contour with **CONTOUR DEF / SEL CONTOUR** or with the OCM shape cycles **127x**.

Closed pockets can also be defined in Cycle **14**.

The machining dimensions, such as milling depth, allowances, and clearance height, can be entered centrally in Cycle **271 OCM CONTOUR DATA** or in the **127x** figure cycles.

CONTOUR DEF / SEL CONTOUR:

In **CONTOUR DEF / SEL CONTOUR**, the first contour can be a pocket or a boundary. The next contours can be programmed as islands or pockets. To program open pockets, use a boundary and an island.

Proceed as follows:

- Program **CONTOUR DEF**
- Define the first contour as a pocket and the second one as an island
- Define Cycle **271 OCM CONTOUR DATA**
- In cycle parameter **Q569**, program the value 1
- > The control will interpret the first contour as an open boundary instead of a pocket. Thus, the open boundary and the island programmed subsequently are combined to form an open pocket.
- Define Cycle **272 OCM ROUGHING**

Programming notes:

- Subsequently defined contours that are outside the first contour will not be considered.
- The first depth of the subcontour is the cycle depth. This is the maximum depth for the programmed contour. Other subcontours cannot be deeper than the cycle depth Therefore, start programming the subcontour with the deepest pocket.

OCM figure cycles:

The figure defined in an OCM figure cycles can be a pocket, an island, or a boundary. Use the Cycles **128x** for programming an island or an open pocket.

Proceed as follows:

- Program a figure using cycles **127x**
- If the first figure will be an island or an open pocket, make sure to program boundary cycle **128x**.
- Define Cycle **272 OCM ROUGHING**

Further information: "OCM cycles for pattern definition", Page 392

Machining operation

When roughing, these cycles allow you to use larger tools for the first roughing passes and then smaller tools to take off the residual material. For finishing, the material roughed out will be taken into consideration.

Example

You defined a Ø20 mm rough-out tool. For roughing, this results in minimum inside radii of 10 mm (cycle parameter **Q578** Radius factor on inside corners will not be taken into account in this example). In the next step, you will finish the contour. For this purpose, define a Ø10 mm finishing cutter. In this case, the minimum inside radii would be 5 mm. Finishing cycles will also consider the previous machining steps, depending on **Q438**, so that the smallest finishing inside radii will be 10 mm. Thus, the finishing cutter will be protected from overload.

Program structure: Machining with OCM cycles

```
0 BEGIN OCM MM
...
12 CONTOUR DEF
...
13 CYCL DEF 271 OCM CONTOUR DATA
...
16 CYCL DEF 272 OCM ROUGHING
...
17 CYCL CALL
...
20 CYCL DEF 273 OCM FINISHING FLOOR
...
21 CYCL CALL
...
24 CYCL DEF 274 OCM FINISHING SIDE
...
25 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM OCM MM
```

14.3.35 Cycle 271 OCM CONTOUR DATA (option 167)

Application

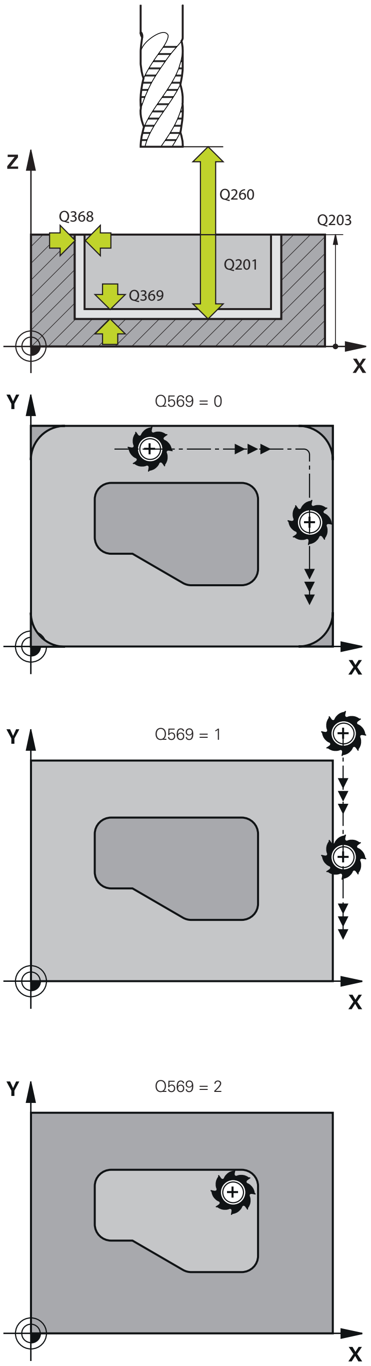
Use Cycle **271 OCM CONTOUR DATA** to program machining data for the contour or the subprograms describing the subcontours. In addition, Cycle **271** enables you to define an open boundary for a pocket.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **271** is DEF-active, which means that it becomes active as soon as it is defined in the NC program.
- The machining data entered in Cycle **271** are valid for Cycles **272** to **274**.

Cycle parameters

Help graphic	Parameter



Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q201 Depth?

Distance between the workpiece surface and the contour floor. This value has an incremental effect.

Input: **-99999.9999...+0**

Q368 Finishing allowance for side?

Finishing allowance in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q369 Finishing allowance for floor?

Finishing allowance for the floor. This value has an incremental effect.

Input: **0...99999.9999**

Q260 Clearance height?

Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q578 Radius factor on inside corners?

The inside radii of the contour are calculated based on the tool radius plus the product of the tool radius times **Q578**.

Input: **0.05...0.99**

Q569 Is the first pocket a boundary?

Define the boundary:

0: The first contour in **CONTOUR DEF** is interpreted as a pocket.

1: The first contour in **CONTOUR DEF** is interpreted as an open boundary. The following contour must be an island

2: The first contour in **CONTOUR DEF** is interpreted as a "bounding block." The following contour must be a pocket

Input: **0**, **1**, **2**

Example

11 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0	;SURFACE COORDINATE ~
Q201=-20	;DEPTH ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q260=+100	;CLEARANCE HEIGHT ~
Q578=+0.2	;INSIDE CORNER FACTOR ~
Q569=+0	;OPEN BOUNDARY

14.3.36 Cycle 272 OCM ROUGHING (option 167)

Application

Use Cycle **272 OCM ROUGHING** to define the technology data for roughing.

In addition, you can use the **OCM** cutting data calculator. The calculated cutting data help to achieve high material removal rates and therefore increase the productivity.

Further information: "OCM Cutting data calculator (option 167)", Page 604

Requirements

Before programming the call of Cycle **272**, you need to program further cycles:

- **CONTOUR DEF / SEL CONTOUR** or Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**

Cycle sequence

1 The tool uses positioning logic to move to the starting point
2 The control determines the starting point automatically based on the pre-positioning and the programmed contour
3 The control moves to the first plunging depth. The plunging depth and the sequence for machining the contours depend on the plunging strategy **Q575**.
   Depending on the definition in Cycle **271 OCM CONTOUR DATA**, parameter **Q569 OPEN BOUNDARY**, the control plunges as follows:
   - **Q569 = 0** or **2**: The tool plunges into the material in a helical or reciprocating movement. The finishing allowance for the side is taken into account.
     Further information: "Plunging behavior with Q569 = 0 or 2", Page 599
   - **Q569 = 1**: The tool plunges vertically outside the open boundary to the first plunging depth
4 After reaching the first plunging depth, the tool mills the contour in an outward or inward direction (depending on **Q569**) at the programmed milling feed rate **Q207**
5 In the next step, the tool is moved to the next plunging depth and repeats the roughing procedure until the programmed contour is completely machined
6 Finally, the tool retracts in the tool axis to the clearance height
7 If there are more contours, the control will repeat the machining process. The control then moves to the contour whose starting point is positioned nearest to the current tool position (depending on the infeed strategy **Q575**).

Plunging behavior with Q569 = 0 or 2

The control generally tries plunging with a helical path. If this is not possible, it tries plunging with a reciprocation movement.

The plunging behavior depends on:

- **Q207 FEED RATE MILLING**
- **Q568 PLUNGING FACTOR**
- **Q575 INFEED STRATEGY**
- **ANGLE**
- **RCUTS**
- $\mathbf{R_{corr}}$ (tool radius **R** + tool oversize **DR**)

Helical:

The helical path is calculated as follows:

$Helicalradius = R_{corr} - RCUTS$

At the end of the plunging movement, the tool executes a semi-circular movement to provide sufficient space for the resulting chips.

Reciprocating

The reciprocation movement is calculated as follows:

$L = 2 * (R_{corr} - RCUTS)$

At the end of the plunging movement, the tool executes a linear movement to provide sufficient space for the resulting chips.

Notes

NOTICE
Caution: Danger to the tool and workpiece! The cycle does not include the corner radius **R2** in the calculation of the milling paths. Even if you use a small overlap factor, residual material may be left over on the contour floor. The residual material can cause damage to the workpiece and the tool during subsequent machining operations! ▶ Run a simulation to verify the machining sequence and the contour ▶ Use tools without a corner radius **R2** where possible

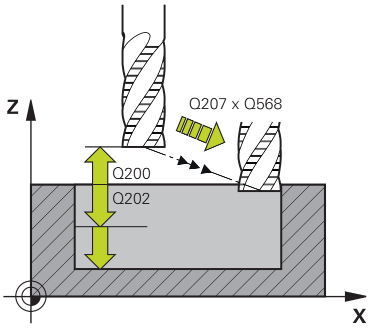
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the plunging depth is larger than **LCUTS**, it will be limited and the control will display a warning.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

> ℹ If required, use a center-cut end mill (ISO 1641).

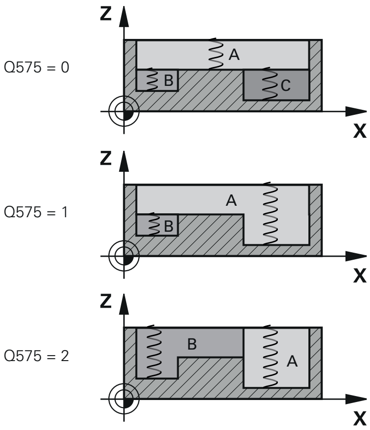
Notes on programming

- **CONTOUR DEF** / **SEL CONTOUR** will reset the tool radius that was used last.
  If you run this machining cycle with **Q438** = –1 after **CONTOUR DEF** / **SEL CONTOUR**, the control assumes that no pre-machining has taken place yet.
- If the path overlap factor **Q370 < 1**, a value of less than 1 is also recommended for the plunging factor **Q579** .

Cycle parameters

Help graphic	Parameter
	**Q202 Plunging depth?** Tool infeed per cut. This value has an incremental effect. Input: **0...99999.9999**
	Q370 Path overlap factor? **Q370** x tool radius = lateral infeed k on a straight line. The control maintains this value as precisely as possible. Input: **0.04...1.99** or **PREDEF**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q568 Factor for plunging feed rate? Factor by which the control reduces the feed rate **Q207** for downfeed into the material. Input: **0.1...1**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between lower edge of tool and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You are able to transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used in Cycle **272** is the rough-out tool (default behavior) **0**: If there was no coarse-roughing, enter the number of a tool with the radius 0. This is usually the tool numbered 0. Input: **-1...+32767.9** or max. **255** characters

Help graphic	Parameter
	Q577 Factor for appr./dept. radius? Factor by which the approach or departure radius will be multiplied. **Q577** is multiplied by the tool radius. This results in an approach and departure radius. Input: **0.15...0.99**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**
	Q576 Spindle speed? Spindle speed in revolutions per minute (rpm) for the roughing tool. **0**: The spindle speed from the **TOOL CALL** block will be used **> 0**: If a value greater than zero is entered, then this spindle speed will be used Input: **0...99999**
	Q579 Factor for plunging speed? Factor by which the control reduces the **SPINDLE SPEED Q576** for downfeed into the material. Input: **0.2...1.5**

Help graphic	Parameter
	**Q575 Infeed strategy (0/1)?** Type of downfeed: **0**: The control machines the contour from top to bottom **1**: The control machines the contour from bottom to top. The control does not always start with the deepest contour. The machining sequence is automatically calculated by the control. The total plunging path is often shorter than with strategy **2**. **2**: The control machines the contour from bottom to top. The control does not always start with the deepest contour. This strategy calculates the machining sequence such that the maximum length of the cutting edge is used. The resulting total plunging path is thus often larger than with strategy **1**. Depending on **Q568**, this may also result in a shorter machining time. Input: **0**, **1**, **2** The total plunging path is the sum of all plunging movements.

Example

11 CYCL DEF 272 OCM ROUGHING ~	
Q202=+5	**;PLUNGING DEPTH ~**
Q370=+0.4	**;TOOL PATH OVERLAP ~**
Q207=+500	**;FEED RATE MILLING ~**
Q568=+0.6	**;PLUNGING FACTOR ~**
Q253=+750	**;F PRE-POSITIONING ~**
Q200=+2	**;SAFETY CLEARANCE ~**
Q438=-1	**;ROUGH-OUT TOOL ~**
Q577=+0.2	**;APPROACH RADIUS FACTOR ~**
Q351=+1	**;CLIMB OR UP-CUT ~**
Q576=+0	**;SPINDLE SPEED ~**
Q579=+1	**;PLUNGING FACTOR S ~**
Q575=+0	**;INFEED STRATEGY**

14.3.37 OCM Cutting data calculator (option 167)

Fundamentals of the OCM cutting data calculator

Introduction

The OCM cutting data calculator is used to determine the Cutting data for Cycle **272 OCM ROUGHING**. These result from the properties of the material and the tool. The calculated cutting data help to achieve high material removal rates and therefore increase the productivity.

In addition, you can use the OCM cutting data calculator to specifically influence the load on the tool via sliders for the mechanical and thermal loads. This allows you to optimize the process reliability, the wear on the tool, and the productivity.

Requirements

> Refer to your machine manual!
>
> In order to capitalize on the calculated Cutting data, you need a sufficiently powerful spindle as well as a stable machine tool.

- The entered values are based on the assumption that the workpiece is firmly clamped in place.
- The entered values are based on the assumption that the tool is seated firmly in its holder.
- The tool being used must be appropriate for the material to be machined.

> In case of large cutting depths and a large angle of twist, strong pulling forces develop in the direction of the tool axis. Make sure to have a sufficient finishing allowance for the floor.

Maintaining the cutting conditions

Use the cutting data only for Cycle **272 OCM ROUGHING**.

Only this cycle ensures that the permissible tool contact angle is not exceeded for the contours to be machined.

Chip removal

NOTICE

Caution: Danger to the tool and workpiece!

If the chips are not removed in an optimum manner, they could get caught in narrow pockets at these high metal removal rates. There is then a risk of tool breakage!

- Ensure that the chips are removed in an optimum manner, as recommended by the OCM cutting data calculator.

Process cooling

The OCM cutting data calculator recommends dry cutting with cooling by compressed air for most materials. The compressed air must be aimed directly at the cutting location. The best method is through the tool holder. If this is not possible, you can also mill with an internal coolant supply.

However, chip removal might not be as efficient when using tools with an internal coolant supply. This can lead to shortened tool life.

Operation

Opening the cutting data calculator

- Select cycle **272 OCM ROUGHING**
- Select **OCM cutting data calculator** in the action bar

Closing the cutting data calculator

Apply

- Select **APPLY**
- > The control applies the determined Cutting data to the intended cycle parameters.
- > The current entries are stored, and are in place when the cutting data calculator is opened again.

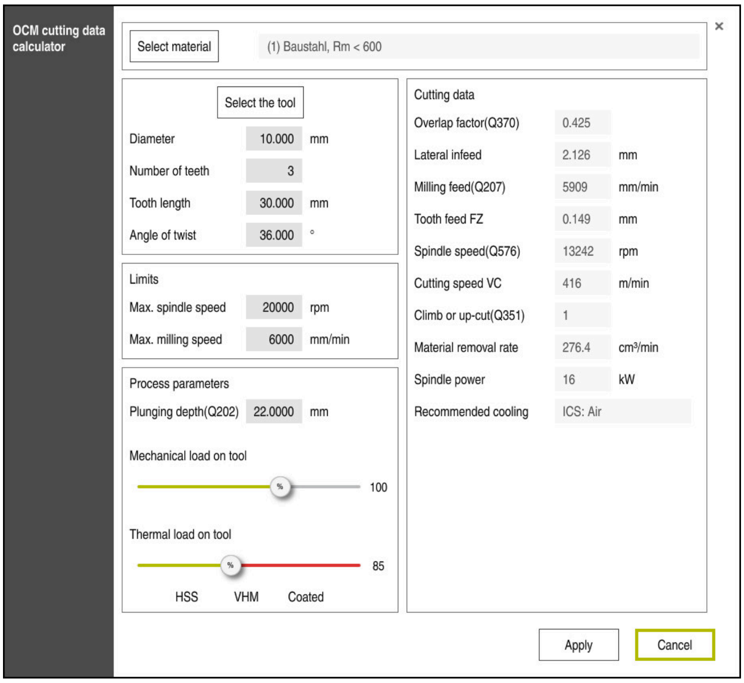
or

Cancel

- Select **Cancel**
- > The current entries are not stored.
- > The control does not apply any values to the cycle.

> The OCM cutting data calculator calculates associated values for these cycle parameters:
> - Plunging depth(Q202)
> - Overlap factor(Q370)
> - Spindle speed(Q576)
> - Climb or up-cut(Q351)
>
> If you use the OCM cutting data calculator, then do not subsequently edit these parameters in the cycle.

Fillable form

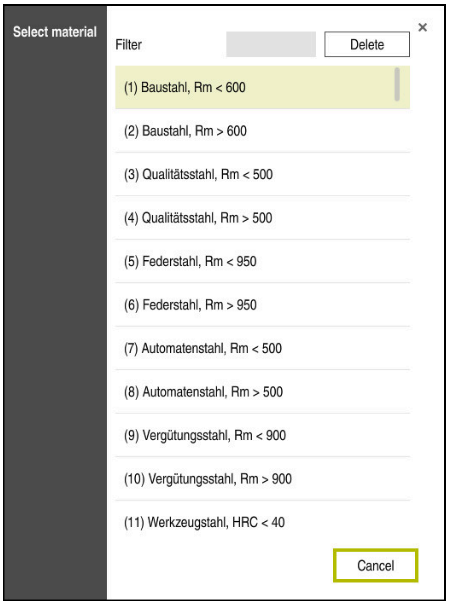


The control uses various colors and symbols in the fillable form:

- Dark gray background: entry required
- Red border of input boxes and information symbols: missing or incorrect entry
- Gray background: no entry possible

> The input field of the workpiece material is highlighted in gray. You can only select it through the selection list. The tool can also be selected through the tool table.

Workpiece material



Proceed as follows to select the workpiece material:

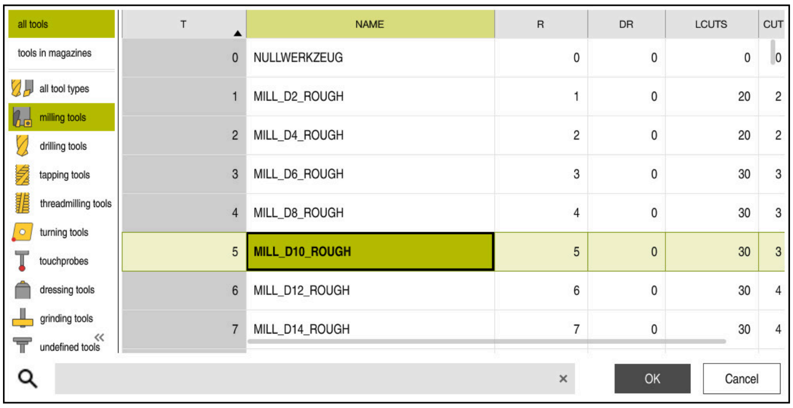
- Select the **Select material** button
- > The control opens a selection list with various types of steel, aluminum, and titanium.
- Select the workpiece material

  or
- Enter a search term in the filter mask
- > The control displays the materials or material groups that were found. Use the **Delete** button to return to the original selection list.

Programming and operating notes:

- If your material is not listed in the table, choose an appropriate material group or a material with similar cutting properties
- You will find the workpiece-material table **ocm.xml** in the **TNC:\system_calcprocess** directory

Tool



You can choose the tool either by selecting it from the tool table **tool.t** or by entering the data manually.

Proceed as follows to select the tool:

- Select the **Select the tool** button
- \> The control opens the active tool table **tool.t**.
- Select the tool

  or
- Enter a tool name or number in the search field
- Confirm with **OK**
- \> The control applies the **Diameter**, the **Number of teeth** and the **Tooth length** from the **tool.t** table.
- Define the **Angle of twist**

Proceed as follows to select the tool:

- Enter the **Diameter**
- Define the **Number of teeth**
- Enter the **Tooth length**
- Define the **Angle of twist**

Input dialog	Description
Diameter	Diameter of the roughing tool in mm Value is applied automatically after the roughing tool has been selected. Input: **1...40**
Number of teeth	Number of teeth of the roughing tool Value is applied automatically after the roughing tool has been selected. Input: **1...10**
Angle of twist	Angle of twist of the roughing tool in ° If there are different angles of twist, then enter the average value. Input: **0...80**

> Programming and operating notes:
> - You can modify the values of the **Diameter**, the **Number of teeth** and the **Tooth length** at any time. The modified value is **not** written to the tool table **tool.t**!
> - You will find the Angle of twist in the description of your tool, for example in the tool catalog of the tool manufacturer.

Limits

For the Limits, you need to define the maximum spindle speed and the maximum milling speed. The calculated Cutting data are then limited to these values.

Input dialog	Description
Max. spindle speed	Maximum spindle speed in rpm permitted by the machine and the clamping situation: Input: **1...99999**
Max. milling speed	Maximum milling speed (feed rate) in mm/min permitted by the machine and the clamping situation: Input: **1...99999**

Process parameters

For the Process parameters, you need to define the Plunging depth(Q202) as well as the mechanical and thermal loads:

Input dialog	Description
Plunging depth(Q202)	Plunging depth (>0 mm to [6 times the tool diameter]) The value from cycle parameter **Q202** is applied when starting the OCM cutting data calculator. Input: **0.001...99999.999**,
Mechanical load on tool	Slider for selection of the mechanical load (the value is normally between 70 % and 100 %) Input: **0%...150%**
Thermal load on tool	Slider for selection of the thermal load Set the slider according to the thermal wear-resistance (coating) of your tool. ■ HSS: low thermal wear-resistance ■ VHM (uncoated or normally-coated solid carbide milling cutters): medium thermal wear-resistance ■ Coated (fully-coated solid carbide milling cutters): high thermal wear-resistance ■ The slider is only effective in the range with a green background. This limiting depends on the maximum spindle speed, the maximum feed rate, and the selected material. ■ If the slider is in the red range, the control will use the maximum permissible value. Input: **0%...200%**

Further information: "Process parameters ", Page 610

Cutting data

The control displays the calculated values in the Cutting data section.

The following Cutting data are applied to the appropriate cycle parameters in addition to the plunging depth **Q202**:

Cutting data:	Applied to cycle parameter:
Overlap factor(Q370)	**Q370** = **TOOL PATH OVERLAP**
Milling feed(Q207) in mm/min	**Q207** = **FEED RATE MILLING**
Spindle speed(Q576) in rpm	**Q576** = **SPINDLE SPEED**
Climb or up-cut(Q351)	**Q351**= **CLIMB OR UP-CUT**

Programming and operating notes:

- The OCM cutting data calculator calculates values only for climb milling **Q351**=+1. For this reason, it always applies **Q351**=+1 to the cycle parameter.
- The OCM cutting data calculator compares the cutting data with the input ranges of the cycle. If the values fall below or exceed the input ranges, the parameter will be highlighted in red in the OCM cutting data calculator. In this case, the cutting data cannot be transferred to the cycle.

The following cutting data is for informational purposes and recommendation:

- Lateral infeed in mm
- Tooth feed FZ in mm
- Cutting speed VC in m/min
- Material removal rate in cm³/min
- Spindle power in kW
- Recommended cooling

These values help you assess whether your machine tool is able to meet the selected cutting conditions.

Process parameters

The two sliders for the mechanical and thermal load have an influence on the process forces and temperatures prevalent on the cutting edges. Higher values increase the metal removal rate, but also lead to a higher load. Moving the sliders makes different process parameters possible.

Maximum material removal rate

For a maximum material removal rate, set the slider for the mechanical load to 100 % and the slider for the thermal load according to the coating of your tool.

If the defined limitations permit it, the cutting data utilize the tool at its mechanical and thermal load capacities. For large tool diameters (D>=16 mm), a very high level of spindle power can be necessary.

For the theoretically expectable spindle power, refer to the cutting data output.

> If the permissible spindle power is exceeded, you can first move the slider for the mechanical load to a lower value. If necessary, you can also reduce the plunging depth (a_p).
>
> Please note that at very high shaft speeds, a spindle running below its rated speed will not attain the rated power.
>
> If you wish to achieve a high material removal rate, you must ensure that chips are removed optimally.

Reduced load and low wear

In order to decrease the mechanical load and the thermal wear, reduce the mechanical load to 70 %. Reduce the thermal load to a value that corresponds to 70 % of the coating of your tool.

These settings utilize the tool in a manner that is mechanically and thermally balanced. In general the tool will then reach its maximum service life. The lower mechanical load makes a smoother process possible that is less subject to vibration.

Achieving an optimum result

If the Cutting data do not lead to a satisfactory cutting process, then different causes might be the reason for this.

Excessively high mechanical load

If there is an excessive mechanical load, you must first reduce the process force.

The following conditions are indications of excessive mechanical load:

- Cutting edges of the tool break
- Shaft of the tool breaks
- Excessive spindle torque or spindle power
- Excessive axial or radial forces on the spindle bearing
- Undesired oscillations or chatter
- Oscillations due to weak clamping
- Oscillations due to long projecting tool

Excessively high thermal load

If there is an excessive thermal load, you must reduce the process temperature.

The following conditions indicate an excessive thermal load on the tool:

- Excessive crater wear at the cutting surface
- The tool glows
- The cutting edges melt (for materials that are very difficult to cut, such as titanium)

Material removal rate is too low

If the machining time is too long and it must be reduced, the material removal rate can be increased by moving both sliders.

If both the machine and the tool still have potential, then it is recommended that the slider for the process temperature be raised to a higher value first. Subsequently, if possible, you can also raise the slider for the process forces to a higher value.

Remedies for problems

The table below provides an overview of possible types of problems as well as countermeasures for them.

Condition	Slider Mechanical load on tool	Slider Thermal load on tool	Miscellaneous
Vibrations (such as weak clamping or tools that project too far)	Decrease	Perhaps increase	Check the clamping
Undesired vibrations or chatter	Decrease	-	
Shaft of tool breaks	Decrease	-	Check the chip removal
Cutting edges of the tool break	Decrease	-	Check the chip removal
Excessive wear	Perhaps increase	Decrease	
The tool glows	Perhaps increase	Decrease	Check the cooling
Machining time is too long	Perhaps increase	Increase this first	
Excessive spindle load	Decrease	-	
Excessive axial force on spindle bearing	Decrease	-	▪ Reduce the plunging depth ▪ Use a tool with a lower angle of twist
Excessive radial force on spindle bearing	Decrease	-	

14.3.38 Cycle 273 OCM FINISHING FLOOR (option 167)

Application

With Cycle **273 OCM FINISHING FLOOR**, you can program finishing with the finishing allowance for the floor programmed in Cycle **271**.

Requirements

Before programming the call of Cycle **273**, you need to program further cycles:

- **CONTOUR DEF / SEL CONTOUR**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **272 OCM ROUGHING**, if applicable

Cycle sequence

1. The control positions the tool to the clearance height at rapid traverse **FMAX**
2. The tool then moves in the tool axis at the feed rate **Q385**
3. The tool smoothly approaches the plane to be machined (on a vertically tangential arc) if there is sufficient room. If there is not enough room, the control moves the tool to depth vertically
4. The tool mills off the material remaining from rough-out (finishing allowance)
5. Finally, the tool retracts in the tool axis to the clearance height

Notes

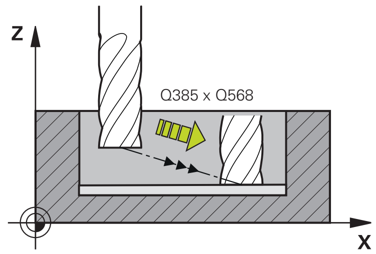
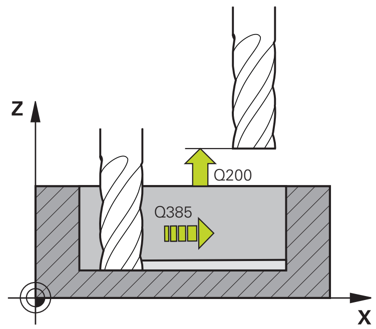
NOTICE
Caution: Danger to the tool and workpiece! The cycle does not include the corner radius **R2** in the calculation of the milling paths. Even if you use a small overlap factor, residual material may be left over on the contour floor. The residual material can cause damage to the workpiece and the tool during subsequent machining operations! ▶ Run a simulation to verify the machining sequence and the contour ▶ Use tools without a corner radius **R2** where possible

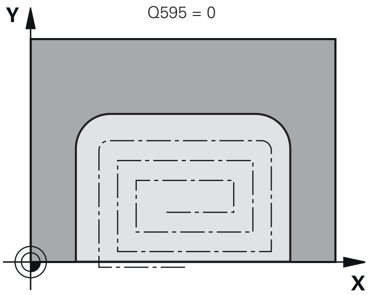
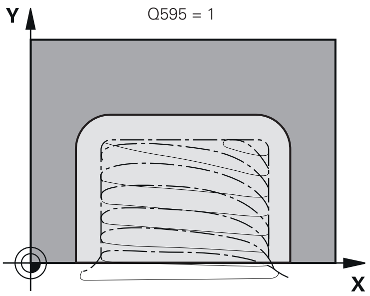
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour.
- For finishing with Cycle **273**, the tool always works in climb milling mode.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Note on programming

- If you use an overlap factor greater than 1, residual material may be left over. Check the contour using the program verification graphics and slightly change the overlap factor, if necessary. This allows another distribution of cuts, which often provides the desired results.

Cycle parameters

Help graphic	Parameter
	Q370 Path overlap factor? **Q370** x tool radius = lateral infeed k. The overlap is considered to be the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners. Input: **0.0001...1.9999** or **PREDEF**
	**Q385 Finishing feed rate?** Traversing speed of the tool in mm/min for floor finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q568 Factor for plunging feed rate? Factor by which the control reduces the feed rate **Q385** for downfeed into the material. Input: **0.1...1**
	**Q253 Feed rate for pre-positioning?** Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between lower edge of tool and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used is the rough-out tool (default behavior). Input: **-1...+32767.9** or max. **255** characters

Help graphic	Parameter
	**Q595 Strategy (0/1)?** Machining strategy for finishing **0**: Equidistant strategy = constant distances between paths **1**: Strategy with constant contact angle Input: **0**, **1**
	**Q577 Factor for appr./dept. radius?** Factor by which the approach or departure radius will be multiplied. **Q577** is multiplied by the tool radius. This results in an approach and departure radius. Input: **0.15...0.99**

Example

```
11 CYCL DEF 273 OCM FINISHING FLOOR ~
    Q370=+1                 ;TOOL PATH OVERLAP ~
    Q385=+500               ;FINISHING FEED RATE ~
    Q568=+0.3               ;PLUNGING FACTOR ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q200=+2                 ;SET-UP CLEARANCE ~
    Q438=-1                 ;ROUGH-OUT TOOL ~
    Q595=+1                 ;STRATEGY ~
    Q577=+0.2               ;APPROACH RADIUS FACTOR
```

14.3.39 Cycle 274 OCM FINISHING SIDE (option 167)

Application

With Cycle **274 OCM FINISHING SIDE**, you can program finishing with the side finishing allowance programmed in Cycle **271**. You can run this cycle in climb or up-cut milling.

Cycle **274** can also be used for contour milling.

Proceed as follows:

- Define the contour to be milled as a single island (without pocket boundary)
- Enter the finishing allowance (**Q368**) in Cycle **271** to be greater than the sum of the finishing allowance **Q14** + radius of the tool being used

Requirements

Before programming the call of Cycle **274**, you need to program further cycles:

- **CONTOUR DEF / SEL CONTOUR**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA**
- Cycle **272 OCM ROUGHING**, if applicable
- Cycle **273 OCM FINISHING FLOOR**, if applicable

Cycle sequence

1. The control positions the tool above the workpiece surface to the starting point for the approach position. This position in the plane results from a tangential arc on which the control moves the tool when approaching the contour
2. The control then moves the tool to the first plunging depth using the feed rate for plunging
3. The tool approaches and moves along the contour helically on a tangential arc until the entire contour is finished. Each subcontour is finished separately
4. Finally, the tool retracts in the tool axis to the clearance height.

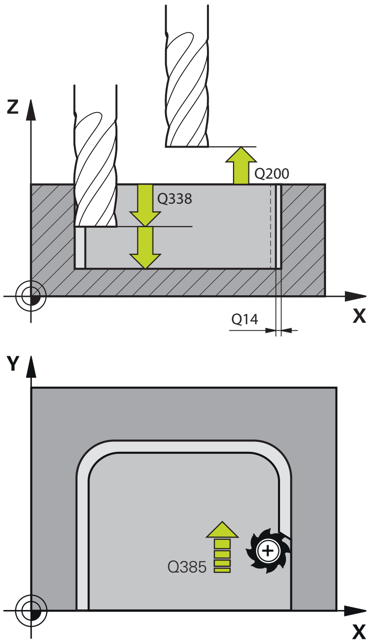
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for finishing. The starting point depends on the available space in the contour and the allowance programmed in Cycle **271**.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- You can execute this cycle using a grinding tool.

Note on programming

- The finishing allowance for the side **Q14** is left over after finishing. It must be smaller than the allowance in Cycle **271**.

Cycle parameters

Help graphic	Parameter
	**Q338 Infeed for finishing?** Tool infeed in the spindle axis per finishing cut. **Q338** = 0: Finishing with a single infeed This value has an incremental effect. Input: **0...99999.9999**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min for side finishing Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min for approaching the starting position. This feed rate will be used below the coordinate surface, but outside the defined material. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between lower edge of tool and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q14 Finishing allowance for side? The finishing allowance for the side **Q14** is left over after finishing. This allowance must be smaller than the allowance in Cycle **271**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used is the rough-out tool (default behavior). Input: **-1...+32767.9** or max. **255** characters
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**

Example

11 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0	;INFEED FOR FINISHING ~
Q385=+500	;FINISHING FEED RATE ~
Q253=+750	;F PRE-POSITIONING ~
Q200=+2	;SET-UP CLEARANCE ~
Q14=+0	;ALLOWANCE FOR SIDE ~
Q438=-1	;ROUGH-OUT TOOL ~
Q351=+1	;CLIMB OR UP-CUT

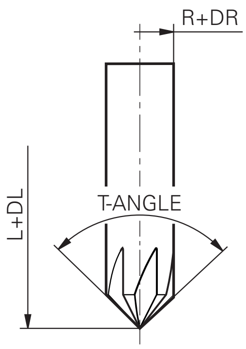
14.3.40 Cycle 277 OCM CHAMFERING (option 167)

Application

Cycle **277 OCM CHAMFERING** enables you to deburr edges of complex contours that you roughed out using OCM cycles.

This cycle considers adjacent contours and boundaries that you called before with Cycle **271 OCM CONTOUR DATA** or the 12xx standard geometric elements.

Requirements



Before the control can execute Cycle **277**, you need to create the tool in the tool table using appropriate parameters:

- **L** + **DL**: Overall length up to the theoretical tip
- **R** + **DR**: Definition of the overall tool radius
- **T-ANGLE**: Point angle of the tool

In addition, you need to program other cycles before programming the call of Cycle **277**:

- **CONTOUR DEF / SEL CONTOUR**, alternatively Cycle **14 CONTOUR**
- Cycle **271 OCM CONTOUR DATA** or the 12xx standard geometric elements
- Cycle **272 OCM ROUGHING**, if applicable
- Cycle **273 OCM FINISHING FLOOR**, if applicable
- Cycle **274 OCM FINISHING SIDE**, if applicable

Cycle sequence

1 The tool uses rapid traverse to move to **Q260 CLEARANCE HEIGHT**. The control takes this information from Cycle **271 OCM CONTOUR DATA** or the 12xx standard geometric elements
2 The tool then moves to the starting point. This point is determined automatically based on the programmed contour.
3 In the next step, the tool moves at **FMAX** to set-up clearance **Q200**
4 Then, the tool plunges vertically to **Q353 DEPTH OF TOOL TIP**
5 The tool approaches the contour in a tangential or vertical movement (depending on the available space). For machining the chamfer, the tool uses the milling feed rate **Q207**
6 Then, the tool is retracted from the contour in a tangential or vertical movement (depending on the available space).
7 If there are several contours, the control positions the tool at clearance height after each contour and then moves it to the next starting point. Steps 3 to 6 are repeated until the programmed contour is completely chamfered
8 At the end of machining, the tool is retracted along the tool axis and moves to **Q260 CLEARANCE HEIGHT**

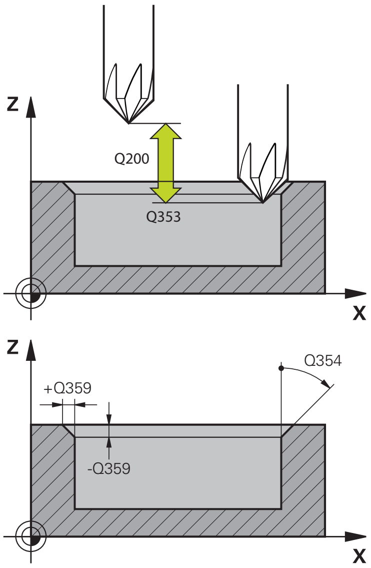
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control automatically calculates the starting point for chamfering. The starting point depends on the available space.
- Measure the length of the tool up to the theoretical tool tip.
- The control monitors the tool radius. Adjacent walls machined with Cycle **271 OCM CONTOUR DATA** or with the **12xx** figure cycles will remain intact.
- Remember that the control does not monitor the theoretical tool tip for collision. In Simulation operating mode, the control always uses the theoretical tool tip for the simulation. This can result, for example for tools without an actual tool tip, in simulated contour damages while the NC program is actually error-free.
- Keep in mind that the effective tool radius must be smaller or equal to the radius of the rough-out tool. Otherwise, the control might not be able to completely chamfer all edges. The effective tool radius is the radius of the cutting length of the tool. The effective tool radius is calculated from **Q353 DEPTH OF TOOL TIP** and **T-ANGLE**.

Note on programming

- If the value of parameter **Q353 DEPTH OF TOOL TIP** is less than the value of parameter **Q359 CHAMFER WIDTH**, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	**Q353 Depth of tool tip?** Distance between theoretical tool tip and workpiece surface coordinate. This value has an incremental effect. Input: **-999.9999...-0.0001**
	Q359 Width of chamfer (-/+)? Width or depth of chamfer: **-**: Depth of chamfer **+**: Width of chamfer This value has an incremental effect. Input: **-999.9999...+999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min for positioning Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q438 or **QS438 Number/name of rough-out tool?** Number or name of the tool that was used by the control to rough out the contour pocket. You can transfer the coarse roughing tool directly from the tool table via the action bar. In addition, you can enter the tool name via the Name in the action bar. The control automatically inserts the closing quotation mark when you exit the input field. **-1**: The control assumes that the tool last used is the rough-out tool (default behavior). Input: **-1...+32767.9** or max. **255** characters
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **-1** = up-cut milling **PREDEF**: The control uses the value of a **GLOBAL DEF** block (If you enter 0, climb milling is performed) Input: **-1**, **0**, **+1** or **PREDEF**

Help graphic	Parameter
	Q354 Angle of chamfer? Angle of the chamfer **0**: The chamfer angle is half the defined **T-ANGLE** from the tool table **> 0**: The chamfer angle is compared to the value of **T-ANGLE** from the tool table. If these two values do not match, the control will display an error message. Input: **0...89**

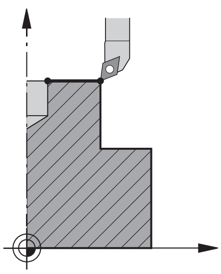
Example

```
11 CYCL DEF 277 OCM CHAMFERING ~
    Q353=-1              ;DEPTH OF TOOL TIP ~
    Q359=+0.2            ;CHAMFER WIDTH ~
    Q207=+500            ;FEED RATE MILLING ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q438=-1              ;ROUGH-OUT TOOL ~
    Q351=+1              ;CLIMB OR UP-CUT ~
    Q354=+0              ;CHAMFER ANGLE
```

14.3.41 Cycle 291 COUPLG.TURNG.INTERP. (option 96)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



Cycle **291 COUPLG.TURNG.INTERP.** couples the tool spindle to the position of the linear axes, or cancels this spindle coupling. With interpolation turning, the cutting edge is oriented to the center of a circle. The center of rotation is defined in the cycle by entering the coordinates **Q216** and **Q217**.

Cycle sequence

Q560=1:

1 The control first performs a spindle stop (**M5**).
2 The control orients the tool spindle to the specified center of rotation. The specified angle for spindle orientation **Q336** is taken into account. If an "ORI" value is given in the tool table, it is also taken into account.
3 The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
4 To terminate the cycle, the coupling must be deactivated by the operator. (With Cycle **291** or end of program/internal stop.)

Q560=0:

1 The control deactivates the spindle coupling.
2 The tool spindle is no longer coupled to the position of the linear axes.
3 The control ends machining with Cycle **291** COUPLG.TURNG.INTERP.
4 If **Q560**=0, parameters **Q336**, **Q216**, **Q217** are not relevant

Notes

This cycle is effective only for machines with servo-controlled spindle.

Your control might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine tool builder for further information.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **291** is CALL-active.
- This cycle can also be used in a tilted working plane.
- Remember that the axis angle must be equal to the tilt angle before the cycle call! Only then can the axis be correctly coupled.
- If Cycle **8 MIRRORING** is active, the control does **not** execute the interpolation turning cycle.
- If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the control does **not** perform the cycle for interpolation turning.

Notes on programming

- Programming of M3/M4 is not required. To describe the circular motions of the linear axes, you can, for example, use **CC** and **C** blocks.
- When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.
- Program outside contours with a radius greater than 0.
- Program inside contours with a radius greater than the tool radius.
- In order to attain high contouring speeds for your machine, define a large tolerance with Cycle **32** before calling the cycle. Program Cycle **32** with HSC filter=1.
- After defining Cycle **291** and **CYCL CALL**, program the operation you wish to perform. To describe the circular motions of the linear axes, you can use linear or polar coordinates, for example.

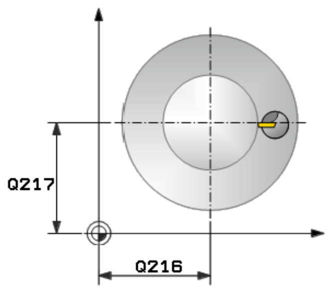
  Further information: "Example: Interpolation turning with Cycle 291", Page 672

Note regarding machine parameters

- In the machine parameter **mStrobeOrient** (no. 201005), the machine manufacturer defines the M function for spindle orientation.
  - If the value is > 0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine manufacturer). The control waits until the oriented spindle stop has been completed.
  - If you enter –1, the control will perform the oriented spindle stop.
  - If you enter 0, no action will be taken.

  The control will, under no circumstances, output **M5** before.

Cycle parameters

Help graphic	Parameter
	Q560 Spindle coupling (0=off, 1=on)? Define whether the tool spindle will be coupled to the position of the linear axes. If spindle coupling is active, the tool's cutting edge is oriented to the center of rotation. **0**: Spindle coupling off **1**: Spindle coupling on Input: **0**, **1**
	Q336 Angle for spindle orientation? The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that one cutting edge is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input: **0...360** Page 625
	**Q216 Center in 1st axis?** Center of rotation in the main axis of the working plane Absolute input: **-99999.9999...99999.9999**
	Q217 Center in 2nd axis? Center of rotation in the secondary axis of the working plane Input: **-99999.9999...+99999.9999**
	Q561 Convert turning tool (0/1) Only relevant if you define the turning tool in the turning tool table (toolturn.trn). This parameter allows you to decide whether the value XL of the turning tool will be interpreted as radius R of a milling tool. **0**: No change; the turning tool is interpreted as described in the turning tool table (toolturn.trn). In this case, you must not use the radius compensation **RR** or **RL**. Furthermore, you must describe the movement of the path of the tool center point **TCP** without spindle coupling when programming. This kind of programming is much more complicated. **1:** The value XL from the turning tool table (toolturn.trn) is interpreted as a radius R of a milling tool table. This makes it possible to use radius compensation **RR** or **RL** when programming your contour. This kind of programming is recommended. Input: **0**, **1**

Example

```
11 CYCL DEF 291 COUPLG.TURNG.INTERP. ~
    Q560=+0                  ;SPINDLE COUPLING ~
    Q336=+0                  ;ANGLE OF SPINDLE ~
    Q216=+50                 ;CENTER IN 1ST AXIS ~
    Q217=+50                 ;CENTER IN 2ND AXIS ~
    Q561=+0                  ;CONVERT FROM TURNING TOOL
```

Defining the tool

Overview

Depending on the entry for parameter **Q560** you can either activate (**Q560**=1) or deactivate (**Q560**=0) the COUPLG.TURNG.INTERP. cycle.

Spindle coupling off, Q560=0

The tool spindle is not coupled to the position of the linear axes.

> **Q560**=0: Disable the **COUPLG.TURNG.INTERP.** cycle!

Spindle coupling on, Q560=1

A turning operation is executed with the tool spindle coupled to the position of the linear axes. If you set the parameter **Q560**=1, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define your turning tool as a milling cutter in the tool table (tool.t). In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). The geometry data of the turning tool are converted to the data of a milling cutter. Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter **Q336** of the cycle. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

  NOTICE

  Danger of collision!

  Collision may occur between the tool holder and workpiece during inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

  - Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**

  You can perform interpolation turning with a milling tool. In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter **Q336**. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

- **Define a turning tool in the turning tool table (toolturn.trn)**

  If you are working with option 50, you can define your turning tool in the turning tool table (toolturn.trn). In this case, the orientation of the spindle to the center of rotation takes place under consideration of tool-specific data, such as the type of machining (TO in the turning tool table), the orientation angle (ORI in the turning tool table), parameter **Q336**, and parameter **Q561**.

Programming and operating notes:

- If you define the turning tool in the turning tool table (toolturn.trn), we recommend working with parameter **Q561**=1. This way, you convert the data of the turning tool into the data of the milling tool, thus greatly facilitating your programming effort. With **Q561**=1 you can use radius compensation **RR** and **RL** when programming. (However, if you program **Q561**=0, then you cannot use radius compensation **RR** and **RL** when describing your contour. Additionally, you must program the movement of the tool center path **TCP** without spindle coupling. This kind of programming is much more complicated!)

  If you programmed parameter **Q561**=1, you must program the following in order to conclude the interpolation turning machining operation:

  - R0, cancels radius compensation
  - Cycle **291** with parameters **Q560**=0 and **Q561**=0, deactivates spindle coupling
  - **CYCL CALL**, for calling Cycle **291**
  - **TOOL CALL** overrides the conversion of parameter **Q561**

  If you programmed parameter **Q561**=1, you may only use the following types of tools:

  - **TYPE**: **ROUGH**, **FINISH**, **BUTTON** with the machining directions **TO**: 1 or 8, **XL**>=0
  - **TYPE**: **ROUGH**, **FINISH**, **BUTTON** with the machining directions **TO**: 7: **XL**<=0

The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	**ORI** + **Q336**
Interpolation turning, inside	7	**ORI** + **Q336** + 180
Interpolation turning, outside	7	**ORI** + **Q336** + 180
Interpolation turning, inside	1	**ORI** + **Q336**
Interpolation turning, outside	8	**ORI** + **Q336**
Interpolation turning, inside	8	**ORI** + **Q336**

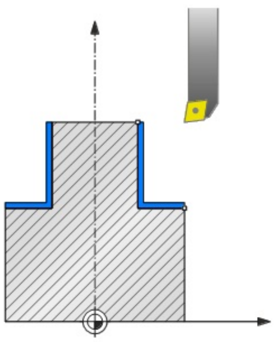
You can use the following tool types for interpolation turning:
- TYPE: ROUGH, with the machining directions TO: 1, 7, 8
- TYPE: FINISH, with the machining directions TO: 1, 7, 8
- TYPE: BUTTON, with the machining directions TO: 1, 7, 8

The following tool types cannot be used for interpolation turning:
- TYPE: ROUGH, with the machining directions TO: 2 to 6
- TYPE: FINISH, with the machining directions TO: 2 to 6
- TYPE: BUTTON, with the machining directions TO: 2 to 6
- TYPE: RECESS
- TYPE: RECTURN
- TYPE: THREAD

14.3.42 Cycle 292 CONTOUR.TURNG.INTRP. (option 96)

Application

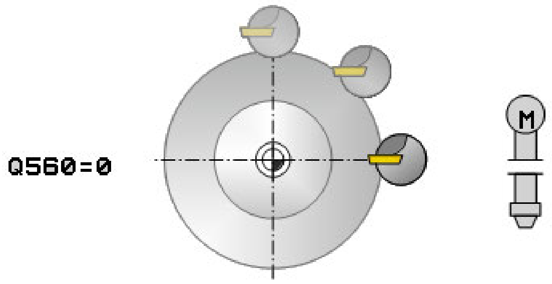
	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.

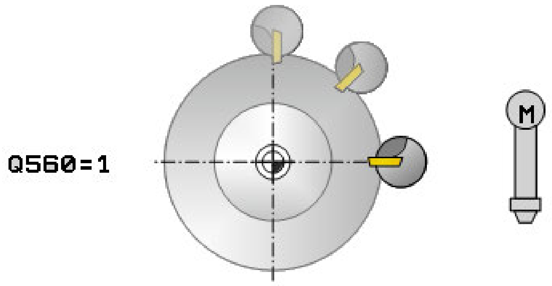


Cycle **292 INTERPOLATION TURNING CONTOUR FINISHING** couples the tool spindle to the positions of the linear axes. This cycle enables you to machine specific rotationally symmetrical contours in the active working plane. You can also run this cycle in the tilted working plane. The center of rotation is the starting point in the working plane at the time the cycle is called. After executing this cycle, the control deactivates the spindle coupling again.

Before using Cycle **292**, you first need to define the desired contour in a subprogram and reference this contour with Cycle **14** or **SEL CONTOUR**. Program the contour either with monotonically decreasing or monotonically increasing coordinates. Undercuts cannot be machined with this cycle. If you enter **Q560**=1, you can turn the contour and the cutting edge is oriented toward the circle center. If you enter **Q560**=0, you can mill the contour and the spindle is not oriented toward the circle center.

Cycle sequence





Cycle Q560=1: Contour turning

1. The control orients the tool spindle to the specified center of rotation. The specified angle **Q336** is taken into account. If an "ORI" value is given in the turning tool table (toolturn.trn), it is also taken into account.
2. The tool spindle is now coupled to the position of the linear axes. The spindle follows the nominal position of the reference axes.
3. The control positions the tool at the contour start radius **Q491**, taking the selected machining operation (inside/outside, **Q529**) and the set-up clearance to the side, **Q357**, into account. The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
4. The control uses the interpolation turning cycle to machine the defined contour. In interpolation turning, the linear axes of the working plane move along a circular path, whereas the spindle axis follows, it is oriented perpendicularly to the surface.
5. At the end point of the contour, the control retracts the tool perpendicularly to set-up clearance.
6. Finally, the control retracts the tool to the clearance height.
7. The control automatically deactivates the coupling of the tool spindle to the linear axes.

Cycle Q560=0: Contour milling

1. The M3/M4 function programmed before the cycle call remains in effect.
2. No spindle stop and **no** spindle orientation will be performed. **Q336** is not taken into account
3. The control positions the tool at the contour start radius **Q491**, taking the selected machining operation (inside/outside, Q529) and the set-up clearance to the side, **Q357**, into account. The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
4. The control machines the defined contour using a rotating spindle (M3/M4). The principal axes of the working plane move along a circular path, whereas the spindle axis does not follow.

5 At the end point of the contour, the control retracts the tool perpendicularly to set-up clearance.
6 Finally, the control retracts the tool to the clearance height.

Notes

This cycle is effective only for machines with servo-controlled spindle.
Your control might monitor the tool to ensure that no positioning movements at feed rate are performed while spindle rotation is off. Contact the machine tool builder for further information.

NOTICE

Danger of collision!

There is a risk of collision between tool and workpiece. The control does not automatically extend the described contour by a set-up clearance! At the beginning of the machining operation, the control positions the tool at rapid traverse FMAX to the contour starting point!

- Program an extension of the contour in the subprogram
- Make sure that there is no material at the contour starting point
- The center of the turning contour is the starting point in the working plane at the time the cycle is called

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The cycle is CALL-active.
- Roughing operations with multiple passes are not possible in this cycle.
- For inside contours, the control checks whether the active tool radius is less than half the diameter at the start of contour **Q491** plus the set-up clearance to the side **Q357**. If the control determines that the tool is too large, the NC program will be canceled.
- Remember that the axis angle must be equal to the tilt angle before the cycle call! Only then can the axis be correctly coupled.
- If Cycle **8 MIRRORING** is active, the control does **not** execute the interpolation turning cycle.
- If Cycle **26 AXIS-SPECIFIC SCALING** is active, and the scaling factor for the axis does not equal 1, the control does **not** perform the cycle for interpolation turning.
- In parameter **Q449 FEED RATE**, you program the feed rate at the starting radius. Keep in mind that the feed rate in the status display is referenced to the **TCP** and may deviate from **Q449**. The control calculates the feed rate in the status display as follows.

  Outside machining **Q529 = 1**

  $$F_{TCP} = Q449 \times \frac{(Q491 + R)}{Q491}$$

  Inside machining **Q529 = 0**

  $$F_{TCP} = Q449 \times \frac{(Q491 - R)}{Q491}$$

Notes on programming

- Program the turning contour without tool radius compensation (RR/RL) and without APPR or DEP movements.
- Please note that it is not possible to define programmed finishing allowances via the **FUNCTION TURNDATA CORR-TCS(WPL)** function. Program a finishing allowance for your contour directly in the cycle or by specifying a tool compensation (DXL, DZL, DRS) in the tool table.
- When programming, remember to use only positive radius values.
- When programming, remember that neither the spindle center nor the indexable insert must be moved into the center of the turning contour.
- Program outside contours with a radius greater than 0.
- Program inside contours with a radius greater than the tool radius.
- In order to attain high contouring speeds for your machine, define a large tolerance with Cycle **32** before calling the cycle. Program Cycle **32** with HSC filter=1.
- If you deactivate the spindle coupling (**Q560 = 0**), you can execute this cycle with polar kinematics. This requires that you clamp the workpiece at the center of the rotary table.

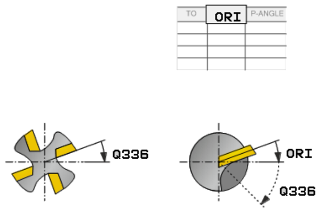
  Further information: "Machining with polar kinematics with FUNCTION POLARKIN", Page 1203

Note regarding machine parameters

- With **Q560**=1, the control does not check whether the cycle is run with a rotating or stationary spindle. (Independent of **CfgGeoCycle** - **displaySpindleError** (no. 201002))
- In the machine parameter **mStrobeOrient** (no. 201005), the machine manufacturer defines the M function for spindle orientation.
  - If the value is > 0, the control executes this M number to perform the oriented spindle stop (PLC function defined by the machine manufacturer). The control waits until the oriented spindle stop has been completed.
  - If you enter –1, the control will perform the oriented spindle stop.
  - If you enter 0, no action will be taken.

  The control will, under no circumstances, output **M5** before.

Cycle parameters

Help graphic	Parameter
	Q560 Spindle coupling (0=off, 1=on)? Define whether the spindle will be coupled or not. **0**: Spindle coupling off (mill the contour) **1**: Spindle coupling on (turn the contour) Input: **0...1**
	**Q336 Angle for spindle orientation?** The control orients the tool to this angle before starting the machining operation. If you work with a milling tool, enter the angle in such a way that one cutting edge is turned towards the center of rotation. If you work with a turning tool, and have defined the value "ORI" in the turning tool table (toolturn.trn), then it is taken into account for the spindle orientation. Input: **0...360**
	Q546 Reverse tool rotation direction? Direction of spindle rotation of the active tool: **3**: Clockwise rotating tool (M3) **4**: Counter-clockwise rotating tool (M4) Input: **3**, **4**
	Q529 Machining operation (0/1)? Define whether an inside or outside contour will be machined: **+1**: Inside machining **0**: Outside machining Input: **0**, **1**
	Q221 Oversize for surface? Allowance in the working plane Input: **0...99.999**
	Q441 Infeed per revolution [mm/rev]? Dimension by which the control moves the tool during one revolution. Input: **0,001...99.999**
	Q449 Feed rate / cutting speed? (mm/min) Feed rate relative to the contour starting point **Q491**. The feed rate of the tool center point path is adjusted depending on the tool radius and **Q529 MACHINING OPERATION**. From these parameters, the control determines the programmed cutting speed at the diameter of the contour starting point. **Q529 = 1**: Feed rate of the tool center point path is reduced for inside machining. **Q529 = 0**: Feed rate of the tool center point path is increased for outside machining. Input: **1...99999** or **FAUTO**

Help graphic	Parameter
	Q491 Contour starting point (radius)? Radius of the contour starting point (e.g., X coordinate, if tool axis is Z). The value has an absolute effect. Input: **0.9999...99999.9999**
	Q357 Safety clearance to the side? Set-up clearance to the side of the workpiece when the tool approaches the first plunging depth. This value has an incremental effect. Input: **0...99999.9999**
	Q445 Clearance height? Absolute height at which collision between tool and workpiece is impossible. The tool retracts to this position at the end of the cycle. Input: **-99999.9999...+99999.9999**

Example

```
11 CYCL DEF 292 CONTOUR.TURNG.INTRP. ~
    Q560=+0                 ;SPINDLE COUPLING ~
    Q336=+0                 ;ANGLE OF SPINDLE ~
    Q546=+3                 ;CHANGE TOOL DIRECTN. ~
    Q529=+0                 ;MACHINING OPERATION ~
    Q221=+0                 ;SURFACE OVERSIZE ~
    Q441=+0.3               ;INFEED ~
    Q449=+2000              ;FEED RATE ~
    Q491=+50                ;CONTOUR START RADIUS ~
    Q357=+2                 ;CLEARANCE TO SIDE ~
    Q445=+50                ;CLEARANCE HEIGHT
```

Machining variants

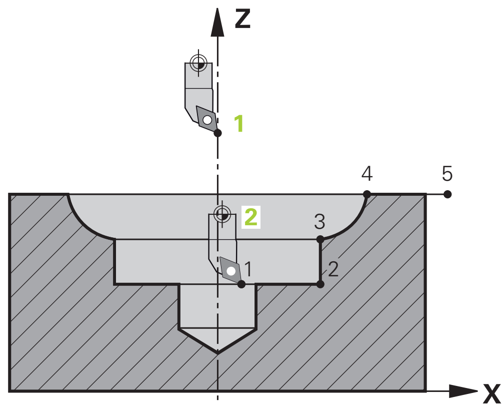
Before using Cycle **292**, you first need to define the desired turning contour in a subprogram and refer to this contour with Cycle **14** or **SEL CONTOUR**. Describe the turning contour on the cross section of a rotationally symmetrical body. Depending on the tool axis, use the following coordinates to define the turning contour:

Tool axis used	Axial coordinate	Radial coordinate
Z	Z	X
X	X	Y
Y	Y	Z

Example: If you are using the tool axis Z, program the turning contour in the axial direction in Z and the radius of the contour in X.

You can use this cycle for inside and outside machining. Some of the notes given in chapter "Notes", Page 630 are illustrated in the following. You will also find an example in "Example: Interpolation Turning Cycle 292", Page 675

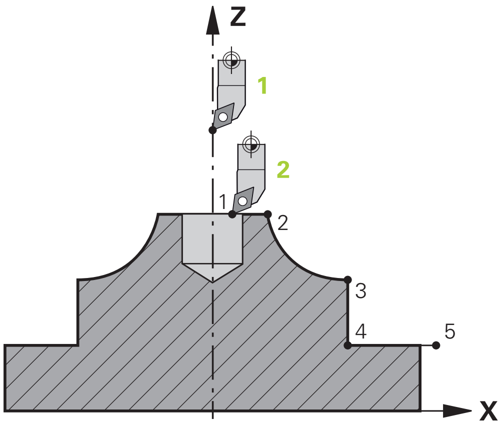
Inside machining



- The center of rotation is the position of the tool in the working plane when the cycle is called (**1**)
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation.** Keep this in mind while describing the contour! (**2**)
- The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
- At the beginning of the machining operation, the control positions the tool to the contour starting point at rapid traverse in the tool axis direction. **Make sure that there is no material at the contour starting point.**

  You also need to take the following into account when programming the inside contour:
  - Program either monotonously increasing radial and axial coordinates (e.g., 1 to 5)
  - Or program monotonously decreasing radial and axial coordinates (e.g., 5 to 1)
  - Program inside contours with a radius greater than the tool radius.

Outside machining



- The center of rotation is the position of the tool in the working plane when the cycle is called (1)
- **Once the cycle has started, do not move the indexable insert or the spindle center into the center of rotation.** Keep this in mind while describing the contour! (2)
- The described contour is not automatically extended by a set-up clearance; you need to program it in the subprogram.
- At the beginning of the machining operation, the control positions the tool to the contour starting point at rapid traverse in the tool axis direction. **Make sure that there is no material at the contour starting point.**

  You also need to take the following into account when programming the outside contour:
  - Program either monotonously increasing radial coordinates and monotonously decreasing axial coordinates (e.g., 1 to 5)
  - Or program monotonously decreasing radial coordinates and monotonously increasing axial coordinates (e.g., 5 to 1)
  - Program outside contours with a radius greater than 0.

Defining the tool

Overview

Depending on the entry for parameter **Q560** you can either mill (**Q560**=0) or turn (**Q560**=1) the contour. For each of the two machining modes, there are different possibilities to define the tool in the tool table. This section describes the different possibilities:

Spindle coupling off, Q560=0

Milling: Define the milling cutter in the tool table as usual by entering the length, radius, toroid cutter radius, etc.

Spindle coupling on, Q560=1

Turning: The geometry data of the turning tool are converted to the data of a milling cutter. You now have the following three possibilities:

- Define a turning tool in the tool table (tool.t) as a milling tool
- Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)
- Define a turning tool in the turning tool table (toolturn.trn)

These three possibilities of defining the tool are described in more detail below:

- **Define a turning tool in the tool table (tool.t) as a milling tool**

  If you are working without option 50, define your turning tool as a milling cutter in the tool table (tool.t). In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align your turning tool to the spindle center. Specify this spindle orientation angle in parameter **Q336** of the cycle. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.

NOTICE

Danger of collision!

Collision may occur between the tool holder and workpiece during inside machining. The tool holder is not monitored. If the tool holder results in a larger rotational diameter than the cutter does, there is a danger of collision.

- Select the tool holder to ensure that it does not result in a larger rotational diameter than the cutter does

- **Define a milling tool in the tool table (tool.t) as a milling tool (for subsequent use as a turning tool)**
  You can perform interpolation turning with a milling tool. In this case, the following data from the tool table are taken into account (including delta values): length (L), radius (R), and corner radius (R2). Align one cutting edge of your milling cutter to the spindle center. Specify this angle in parameter **Q336**. For outside machining, the spindle orientation equals the value in **Q336**, and for inside machining, the spindle orientation equals **Q336**+180.
- **Define a turning tool in the turning tool table (toolturn.trn)**
  If you are working with option 50, you can define your turning tool in the turning tool table (toolturn.trn). In this case, the orientation of the spindle to the center of rotation takes place under consideration of tool-specific data, such as the type of machining (TO in the turning tool table), the orientation angle (ORI in the turning tool table) and parameter **Q336**.
  The spindle orientation is calculated as follows:

Machining	TO	Spindle orientation
Interpolation turning, outside	1	ORI + **Q336**
Interpolation turning, inside	7	ORI + **Q336** + 180
Interpolation turning, outside	7	ORI + **Q336** + 180
Interpolation turning, inside	1	ORI + **Q336**
Interpolation turning, outside	8,9	ORI + **Q336**
Interpolation turning, inside	8,9	ORI + **Q336**

You can use the following tool types for interpolation turning:

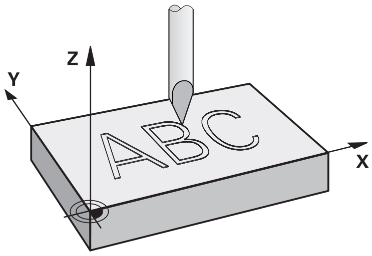
- **TYPE**: **ROUGH**, with the machining directions **TO**: 1 or 7
- **TYPE**: **FINISH**, with the machining directions **TO**: 1 or 7
- **TYPE**: **BUTTON**, with the machining directions **TO**: 1 or 7

The following tool types cannot be used for interpolation turning:

- **TYPE**: **ROUGH**, with the machining directions **TO**: 2 to 6
- **TYPE**: **FINISH**, with the machining directions **TO**: 2 to 6
- **TYPE**: **BUTTON**, with the machining directions **TO**: 2 to 6
- **TYPE**: **RECESS**
- **TYPE**: **RECTURN**
- **TYPE**: **THREAD**

14.3.43 Cycle 225 ENGRAVING

Application



This cycle is used to engrave texts on a flat surface of the workpiece. You can arrange the texts in a straight line or along an arc.

Cycle sequence

1 If the tool is beneath **Q204 2ND SET-UP CLEARANCE**, the control will first move to the value from **Q204**.
2 The control positions the tool in the working plane to the starting point of the first character.
3 The control engraves the text.
   - If **Q202 MAX. PLUNGING DEPTH** is greater than **Q201 DEPTH**, the control will engrave each character in a single infeed motion.
   - If **Q202 MAX. PLUNGING DEPTH** is less than **Q201 DEPTH**, the control will engrave each character in several infeed motions. The control will always complete the milling of a character before machining the next one.
4 After the control has engraved a character, it retracts the tool to the set-up clearance **Q200** above the workpiece surface.
5 The process steps 2 and 3 are repeated for all characters to be engraved.
6 Finally, the control retracts the tool to 2nd set-up clearance **Q204**.

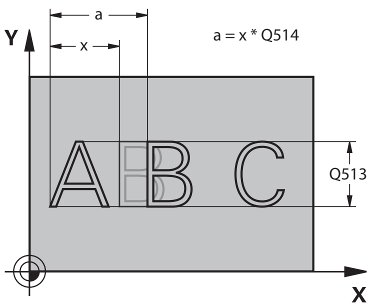
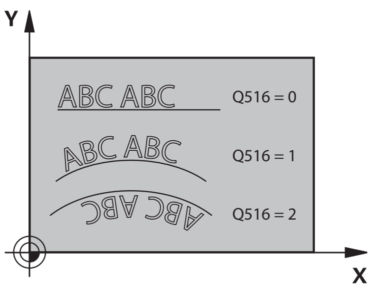
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

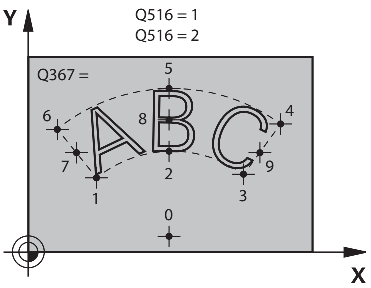
Notes on programming

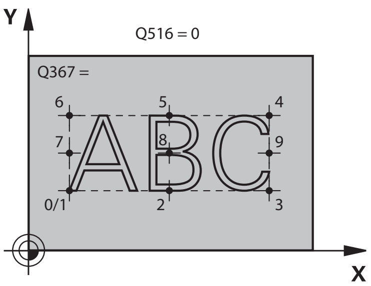
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The text to be engraved can also be transferred with a string variable (**QS**).
- Parameter **Q347** influences the rotational position of the letters.
  If **Q374** = 0° to 180°, the characters are engraved from left to right.
  If **Q374** is greater than 180°, the direction of engraving is reversed.

Cycle parameters

Help graphic	Parameter
	QS500 Engraving text? Text to be engraved within quotation marks. Assignment of a string variable through the **Q** key of the numerical keypad. The **Q** key on the alphabetic keyboard represents normal text input. Input: Max. **255** characters
	**Q513 Character height?** Height of the characters to be engraved in mm Input: **0...999.999**
	Q514 Character spacing factor? The font used is a proportional font. Each character has its own width, which is engraved correspondingly by the control if you program **Q514** = 0. If **Q514** is not equal to 0, the control scales the space between the characters. Input: **0...10**
	Q515 Font? By default, the control uses the **DeJaVuSans** font.
	**Q516 Text on a line/on an arc(0-2)?** **0**: Engrave text in a straight line **1**: Engrave text along an arc **2**: Engrave text along the inside of a circular arc (circumferentially; not necessarily legible from below) Input: **0**, **1**, **2**
	Q374 Angle of rotation? Center angle if the text is arranged on an arc. Engraving angle when text is in a straight line. Input: **-360.000...+360.000**
	Q517 Radius of text on an arc? Radius of the arc in mm on which the control will engrave the text. Input: **0...99999.9999**
	Q207 Feed rate for milling? Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q201 Depth? Distance between workpiece surface and engraving floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q206 Feed rate for plunging? Tool traversing speed in mm/min during plunging Input: **0...99999.999** or **FAUTO**, **FU**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**





Q367 Reference for text position (0-6)?

Enter the reference for the position of the text here. Depending on whether the text will be engraved along a circular arc or in a straight line (parameter **Q516**), the following values can be entered:

Circle	Straight line
0 = Circle center	0 = Bottom left
1 = Bottom left	1 = Bottom left
2 = Bottom center	2 = Bottom center
3 = Bottom right	3 = Bottom right
4 = Top right	4 = Top right
5 = Top center	5 = Top center
6 = Top left	6 = Top left
7 = Center left	7 = Center left
8 = Center of text	8 = Center of text
9 = Center right	9 = Center right

Input: **0...9**

Help graphic	Parameter
	Q574 Maximum text length? Enter the maximum text length. The control also takes into account parameter **Q513** Character height. If **Q513 = 0**, the control engraves the text over exactly the length indicated in parameter **Q574**. The character height will be scaled accordingly. If **Q513 > 0**, the control checks whether the actual text length exceeds the maximum text length entered in **Q574**. If that is the case, the control displays an error message. Input: **0...999.999**
	Q202 Maximum plunging depth? Maximum infeed depth per cut. The machining operation is performed in several steps if this value is less than **Q201**. Input: **0...99999.9999**

Example

```
11 CYCL DEF 225 ENGRAVING ~
    QS500=""            ;ENGRAVING TEXT ~
    Q513=+10            ;CHARACTER HEIGHT ~
    Q514=+0             ;SPACE FACTOR ~
    Q515=+0             ;FONT ~
    Q516=+0             ;TEXT ARRANGEMENT ~
    Q374=+0             ;ANGLE OF ROTATION ~
    Q517=+50            ;CIRCLE RADIUS ~
    Q207=+500           ;FEED RATE MILLING ~
    Q201=-2             ;DEPTH ~
    Q206=+150           ;FEED RATE FOR PLNGNG ~
    Q200=+2             ;SET-UP CLEARANCE ~
    Q203=+0             ;SURFACE COORDINATE ~
    Q204=+50            ;2ND SET-UP CLEARANCE ~
    Q367=+0             ;TEXT POSITION ~
    Q574=+0             ;TEXT LENGTH ~
    Q202=+0             ;MAX. PLUNGING DEPTH
```

Allowed engraving characters

The following special characters are allowed in addition to lowercase letters, uppercase letters and numbers: **! # $ % & ‘ ( ) * + , - . / : ; < = > ? @ [\ ] _ ß CE**

> The control uses the special characters **%** and **** for special functions. If you want to engrave these characters, enter them twice in the text to be engraved e. g. **%%**).

When engraving German umlauts, ß, ø, @, or the CE character, enter the character % before the character to be engraved:

Input	Algebraic sign
%ae	ä
%oe	ö
%ue	ü
%AE	Ä
%OE	Ö
%UE	Ü
%ss	ß
%D	ø
%at	@
%CE	CE

Characters that cannot be printed

Apart from text, you can also define certain non-printable characters for formatting purposes. Enter the special character **** before the non-printable characters.

The following formatting possibilities are available:

Input	Character
\n	Line break
\t	Horizontal tab (the tab width is permanently set to eight characters)
\v	Vertical tab (the tab width is permanently set to one line)

Engraving system variables

In addition to the standard characters, you can engrave the contents of certain system variables. Precede the system variable with **%**.

You can also engrave the current date, the current time, or the current calendar week. Do do so, enter **%time<x>**. **<x>** defines the format, e.g. 08 for DD.MM.YYYY. (Identical to the **SYSSTR ID10321** function)

> Keep in mind that you must enter a leading 0 when entering the date formats 1 to 9, e.g. **%time08**.

Input	Character
%time00	DD.MM.YYYY hh:mm:ss
%time01	D.MM.YYYY h:mm:ss
%time02	D.MM.YYYY h:mm
%time03	D.MM.YY h:mm
%time04	YYYY-MM-DD hh:mm:ss
%time05	YYYY-MM-DD hh:mm
%time06	YYYY-MM-DD h:mm
%time07	YY-MM-DD h:mm
%time08	DD.MM.YYYY
%time09	D.MM.YYYY
%time10	D.MM.YY
%time11	YYYY-MM-DD
%time12	YY-MM-DD
%time13	hh:mm:ss
%time14	h:mm:ss
%time15	h:mm
%time99	ISO 8601 calendar week

> Properties:
> - It comprises seven days
> - It begins with Monday
> - It is numbered sequentially
> - The first calendar week (week 01) is the week with the first Thursday of the Gregorian year.

Engraving the name and path of an NC program

Use Cycle **225** to engrave the name and path of an NC program.

Define Cycle **225** as usual. Precede the engraved text with **%**.

It is possible to engrave the name or path of an active or called NC program. For this purpose, define **%main<x>** or **%prog<x>**. (Identical to the **SYSSTR ID10010 NR1/2** function)

The following formatting possibilities are available:

Input	Meaning	Example
%main0	Full path of the active NC program	**TNC:\MILL.h**
%main1	Path to the directory of the active NC program	**TNC:**
%main2	Name of the active NC program	**MILL**
%main3	File type of the active NC program	**.H**
%prog0	Full path of the called NC program	**TNC:\HOUSE.h**
%prog1	Path to the directory of the called NC program	**TNC:**
%prog2	Name of the called NC program	**HOUSE**
%prog3	File type of the active NC program	**.H**

Engraving the counter reading

Cycle **225** allows you to engrave the current counter reading (provided on the PGM tab of the **Status** work status).

To do so, program Cycle **225** as usual and enter the text to be engraved, for example: **%count2**

The number after **%count** indicates how many digits the control will engrave. The maximum is nine digits.

Example: If you program **%count9** in the cycle with a momentary counter reading of 3, the control will engrave the following: 000000003

Further information: "Defining counters with FUNCTION COUNT", Page 1308

Operating notes

- In Simulation, the control simulates only the counter reading that you have specified directly in the NC program. The counter reading from the program run is not taken into account.

14.3.44 Cycle 232 FACE MILLING

Application

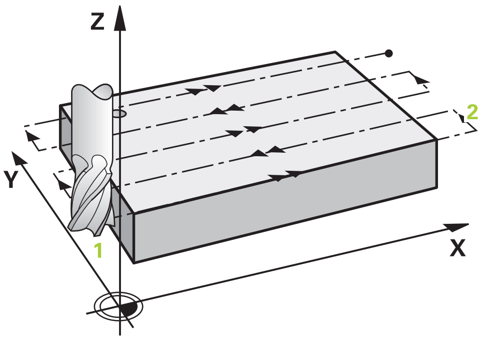
With Cycle **232**, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. Three machining strategies are available:

- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2:** Line-by-line machining, retraction and stepover at the positioning feed rate

Cycle sequence

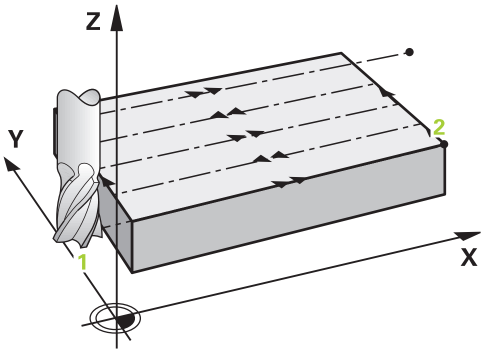
1. From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** using positioning logic: If the current position in the spindle axis is further away from the workpiece than the 2nd set-up clearance, the control positions the tool first in the working plane and then in the spindle axis. Otherwise, it first moves it to 2nd set-up clearance and then in the working plane. The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.
2. The tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the control.

Strategy Q389=0



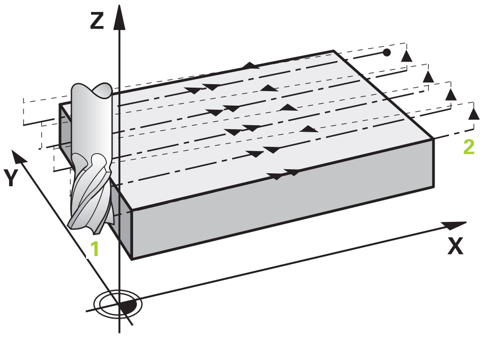
3. The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **outside** the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.
4. The control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5. The tool then moves back in the direction of the starting point **1**.
6. The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7. In order to avoid non-productive motions, the surface is then machined in reverse direction.
8. The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9. At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Strategy Q389=1



3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies **at the edge** of the surface. The control calculates the end point from the programmed starting point, the programmed length and the tool radius.
4 The control offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**. The motion to the next pass again occurs at the edge of the workpiece.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Strategy Q389=2



3 The tool subsequently advances at the programmed feed rate for milling to the end point **2**. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed set-up clearance to the side and the tool radius.
4 The control positions the tool in the spindle axis to the set-up clearance above the current infeed depth, and then moves it at the pre-positioning feed rate directly back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then returns to the current infeed depth and moves in the direction of end point **2**
6 The process is repeated until the programmed surface has been machined completely. At the end of the last pass, the tool plunges to the next machining depth.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at **FMAX** to the 2nd set-up clearance.

Notes

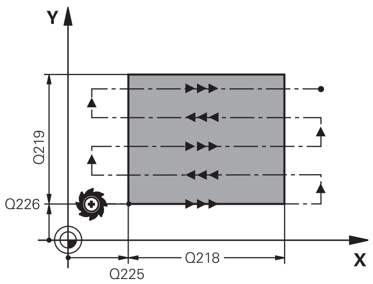
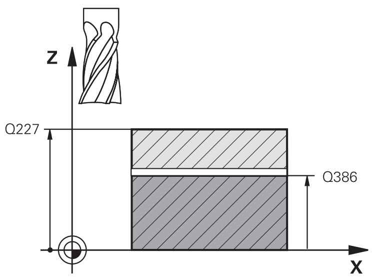
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

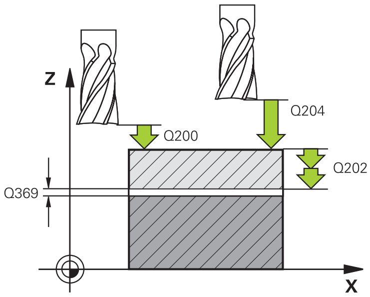
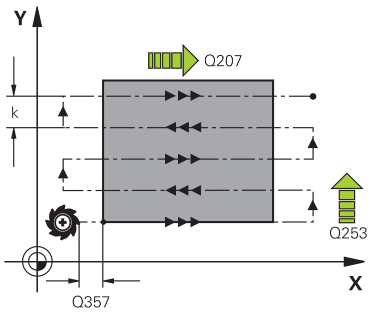
Notes on programming

- If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).
- Program **Q227** greater than **Q386**. The control will otherwise display an error message.

i	Enter **Q204 2ND SET-UP CLEARANCE** in such a way that no collision with the workpiece or the fixtures can occur.

Cycle parameters

Help graphic	Parameter
	Q389 Machining strategy (0/1/2)? Define how the control will machine the surface: **0**: Meander machining, stepover at positioning feed rate outside the surface to be machined **1**: Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined **2**: Line-by-line machining, retraction and stepover at the positioning feed rate Input: **0**, **1**, **2**
	**Q225 Starting point in 1st axis?** Define the starting point coordinate of the surface to be machined in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q226 Starting point in 2nd axis? Define the starting point coordinate of the surface to be machined in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q227 Starting point in 3rd axis? Coordinate of the workpiece surface used to calculate the infeeds. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q386 End point in 3rd axis? Coordinate in the spindle axis on which the surface will be face-milled. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q218 First side length? Length of the surface to be machined in the main axis of the working plane. Use the algebraic sign to specify the direction of the first milling path referenced to the **starting point in the 1st axis**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q219 Second side length? Length of the surface to be machined in the secondary axis of the working plane. Use algebraic signs to specify the direction of the first cross feed referenced to the **STARTNG PNT 2ND AXIS**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	**Q202 Maximum plunging depth?** **Maximum** infeed per cut. The control calculates the actual plunging depth from the difference between the end point and starting point in the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time. This value has an incremental effect. Input: **0...99999.9999**
	Q369 Finishing allowance for floor? Value used for the last infeed. This value has an incremental effect. Input: **0...99999.9999**
	Q370 Max. path overlap factor? Maximum stepover factor k. The control calculates the actual stepover from the second side length (**Q219**) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. cutter radius when using a face-milling cutter), the control reduces the stepover accordingly. Input: **0.001...1.999**
	**Q207 Feed rate for milling?** Traversing speed of the tool in mm/min for milling Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q385 Finishing feed rate? Traversing speed of the tool in mm/min while milling the last infeed Input: **0...99999.999** or **FAUTO**, **FU**, **FZ**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely inside the material (**Q389**=1), the control uses the cross feed rate for milling **Q207**. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy **Q389** = 2, the control moves the tool to set-up clearance above the current plunging depth to the starting point of the next pass. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

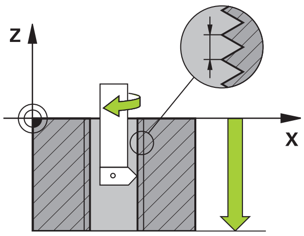
Help graphic	Parameter
	Q357 Safety clearance to the side? Parameter **Q357** influences the following situations: **Approaching the first infeed depth: Q357** is the lateral distance from the tool to the workpiece. **Roughing with the Q389 = 0 to 3 roughing strategies:** The surface to be machined is extended in **Q350 MILLING DIRECTION** by the value from **Q357** if no limit has been set in that direction. **Side finishing:** The paths are extended by **Q357** in the **Q350 MILLING DIRECTION**. Input: **0...99999.9999**
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 CYCL DEF 232 FACE MILLING ~
    Q389=+2          ;STRATEGY ~
    Q225=+0          ;STARTNG PNT 1ST AXIS ~
    Q226=+0          ;STARTNG PNT 2ND AXIS ~
    Q227=+2.5        ;STARTNG PNT 3RD AXIS ~
    Q386=0           ;END POINT 3RD AXIS ~
    Q218=+150        ;FIRST SIDE LENGTH ~
    Q219=+75         ;2ND SIDE LENGTH ~
    Q202=+5          ;MAX. PLUNGING DEPTH ~
    Q369=+0          ;ALLOWANCE FOR FLOOR ~
    Q370=+1          ;MAX. OVERLAP ~
    Q207=+500        ;FEED RATE MILLING ~
    Q385=+500        ;FINISHING FEED RATE ~
    Q253=+750        ;F PRE-POSITIONING ~
    Q200=+2          ;SET-UP CLEARANCE ~
    Q357=+2          ;CLEARANCE TO SIDE ~
    Q204=+50         ;2ND SET-UP CLEARANCE
```

14.3.45 Cycle 18 THREAD CUTTING

Application



Cycle **18 THREAD CUTTING** moves the tool with servo-controlled spindle from the momentary position with active speed to the specified depth. As soon as it reaches the end of thread, spindle rotation is stopped. Approach and departure movements must be programmed separately.

Notes

NOTICE

Danger of collision!

If you do not program a pre-positioning step before programming the call of Cycle **18**, a collision might occur. Cycle **18** does not perform any approach or departure movements.

- Pre-position the tool before the start of the cycle.
- The tool moves from the current position to the entered depth after the cycle is called

NOTICE

Danger of collision!

If the spindle was switched on before the start of this cycle, Cycle **18** will switch it off and the cycle will execute with a stationary spindle! At the end, Cycle **18** will switch the spindle on again if it was on before the start of the cycle.

- Before starting this cycle, be sure to program a spindle stop! (For example with **M5**)
- At the end of Cycle **18**, the control restores the spindle to its state at cycle start. This means that if the spindle was switched off before this cycle, the control will switch it off again at the end of Cycle **18**.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

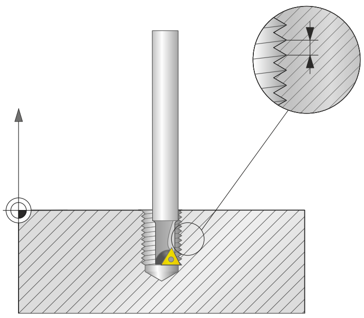
Notes on programming

- Before calling this cycle, program a spindle stop (for example with M5). The control automatically activates spindle rotation at the start of the cycle and deactivates it at the end.
- The algebraic sign for the cycle parameter "thread depth" determines the working direction.

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
  - **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (spindle speed override is not active); the control then adjusts the spindle speed as required
  - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
  - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
  - **limitSpindleSpeed** (no. 113604): Spindle speed limit
    True: At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time.
    False: Limiting not active

Cycle parameters

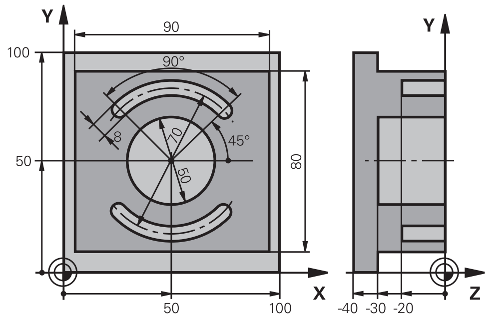
Help graphic	Parameter
	**Total hole depth?** Enter the thread depth relative to the current position. This value has an incremental effect. Input: **-999999999...+999999999**
	Thread pitch? Enter the thread pitch. The algebraic sign entered here differentiates between right-hand and left-hand threads: **+** = Right-hand thread (M3 with negative hole depth) **-** = Left-hand thread (M4 with negative hole depth) Input: **-99.9999...+99.9999**

Example

```
11 CYCL DEF 18.0 THREAD CUTTING
12 CYCL DEF 18.1 DEPTH-20
13 CYCL DEF 18.2 PITCH+1
```

14.3.46 Programming examples

Example: Milling pockets, studs and slots

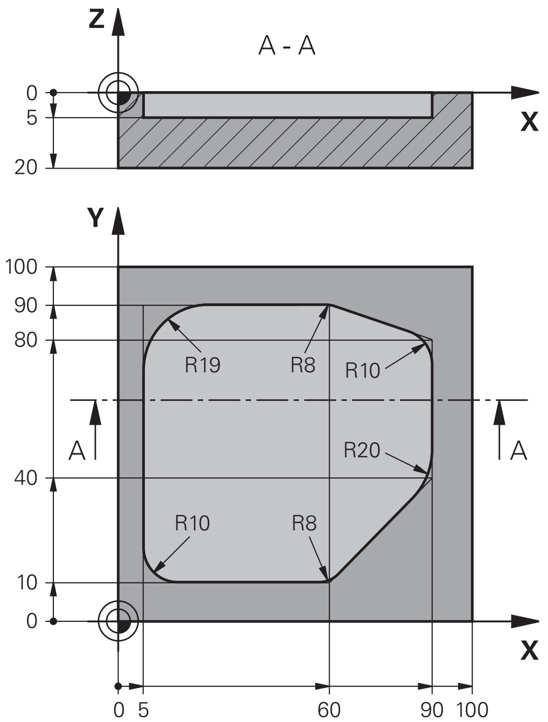


0 BEGIN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 6 Z S3500	; Tool call: roughing/finishing
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD ~	
Q218=+90 ;FIRST SIDE LENGTH ~	
Q424=+100 ;WORKPC. BLANK SIDE 1 ~	
Q219=+80 ;2ND SIDE LENGTH ~	
Q425=+100 ;WORKPC. BLANK SIDE 2 ~	
Q220=+0 ;CORNER RADIUS ~	
Q368=+0 ;ALLOWANCE FOR SIDE ~	
Q224=+0 ;ANGLE OF ROTATION ~	
Q367=+0 ;STUD POSITION ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-30 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+20 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q437=+0 ;APPROACH POSITION ~	
Q215=+0 ;MACHINING OPERATION ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q338=+10 ;INFEED FOR FINISHING ~	
Q385=+500 ;FINISHING FEED RATE	
6 L X+50 Y+50 R0 FMAX M99	; Cycle call for outside machining
7 CYCL DEF 252 CIRCULAR POCKET ~	
Q215=+0 ;MACHINING OPERATION ~	

Q223=+50 ;CIRCLE DIAMETER ~	
Q368=+0.2 ;ALLOWANCE FOR SIDE ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-30 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q338=+5 ;INFEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q366=+1 ;PLUNGE ~	
Q385=+750 ;FINISHING FEED RATE ~	
Q439=+0 ;FEED RATE REFERENCE	
8 L X+50 Y+50 R0 FMAX M99	; Cycle call for circular pocket
9 TOOL CALL 3 Z S5000	; Tool call: slot milling cutter
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 254 CIRCULAR SLOT ~	
Q215=+0 ;MACHINING OPERATION ~	
Q219=+8 ;SLOT WIDTH ~	
Q368=+0.2 ;ALLOWANCE FOR SIDE ~	
Q375=+70 ;PITCH CIRCLE DIAMETR ~	
Q367=+0 ;REF. SLOT POSITION ~	
Q216=+50 ;CENTER IN 1ST AXIS ~	
Q217=+50 ;CENTER IN 2ND AXIS ~	
Q376=+45 ;STARTING ANGLE ~	
Q248=+90 ;ANGULAR LENGTH ~	
Q378=+180 ;STEPPING ANGLE ~	
Q377=+2 ;NR OF REPETITIONS ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-20 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q338=+5 ;INFEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q366=+2 ;PLUNGE ~	
Q385=+500 ;FINISHING FEED RATE ~	

Q439=+0 ;FEED RATE REFERENCE	
12 CYCL CALL	; Cycle call for slots
13 L Z+100 R0 FMAX	; Retract the tool, end program
14 M30	
15 END PGM C210 MM	

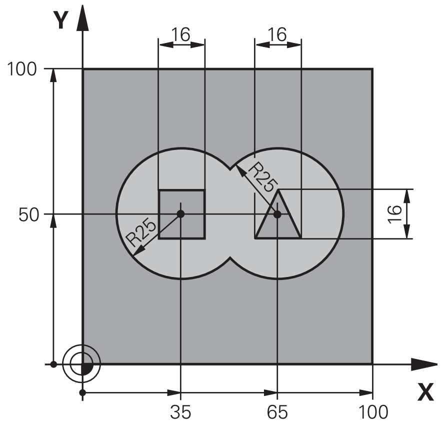
Example: Roughing-out and fine-roughing a pocket with SL Cycles



0 BEGIN PGM 1078634 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 15 Z S4500	; Tool call: coarse roughing tool (diameter: 30)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL 1	
7 CYCL DEF 20 CONTOUR DATA ~	
Q1=-5 ;MILLING DEPTH ~	
Q2=+1 ;TOOL PATH OVERLAP ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q4=+0 ;ALLOWANCE FOR FLOOR ~	
Q5=+0 ;SURFACE COORDINATE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q7=+50 ;CLEARANCE HEIGHT ~	
Q8=+0.2 ;ROUNDING RADIUS ~	
Q9=+1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q12=+500 ;FEED RATE F. ROUGHNG ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q19=+200 ;FEED RATE FOR RECIP. ~	

Q208=+99999 ;RETRACTION FEED RATE ~	
Q401=+90 ;FEED RATE FACTOR ~	
Q404=+1 ;FINE ROUGH STRATEGY	
9 CYCL CALL	; Cycle call: coarse roughing
10 L Z+200 R0 FMAX	; Retract the tool
11 TOOL CALL 4 Z S3000	; Tool call: fine roughing tool (diameter: 8)
12 L Z+100 R0 FMAX M3	
13 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q12=+500 ;FEED RATE F. ROUGHNG ~	
Q18=+15 ;COARSE ROUGHING TOOL ~	
Q19=+200 ;FEED RATE FOR RECIP. ~	
Q208=+99999 ;RETRACTION FEED RATE ~	
Q401=+90 ;FEED RATE FACTOR ~	
Q404=+1 ;FINE ROUGH STRATEGY	
14 CYCL CALL	; Cycle call: fine roughing
15 L Z+200 R0 FMAX	; Retract the tool
16 M30	; End of program
17 LBL 1	; Contour subprogram
18 L X+5 Y+50 RR	
19 L Y+90	
20 RND R19	
21 L X+60	
22 RND R8	
23 L X+90 Y+80	
24 RND R10	
25 L Y+40	
26 RND R20	
27 L X+60 Y+10	
28 RND R8	
29 L X+5	
30 RND R10	
31 L X+5 Y+50	
32 LBL 0	
33 END PGM 1078634 MM	

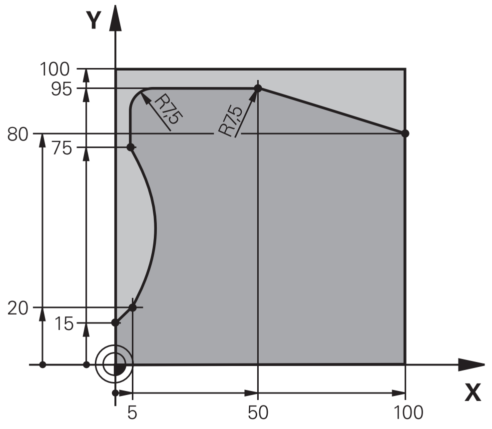
Example: Pilot drilling, roughing and finishing overlapping contours with SL Cycles



0 BEGIN PGM 2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 204 Z S2500	; Tool call: drill (diameter: 12)
4 L Z+250 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1 /2 /3 /4	
7 CYCL DEF 20 CONTOUR DATA ~	
Q1=-20 ;MILLING DEPTH ~	
Q2=+1 ;TOOL PATH OVERLAP ~	
Q3=+0.5 ;ALLOWANCE FOR SIDE ~	
Q4=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q5=+0 ;SURFACE COORDINATE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q7=+100 ;CLEARANCE HEIGHT ~	
Q8=+0.1 ;ROUNDING RADIUS ~	
Q9=-1 ;ROTATIONAL DIRECTION	
8 CYCL DEF 21 PILOT DRILLING ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+150 ;FEED RATE FOR PLNGNG ~	
Q13=+0 ;ROUGH-OUT TOOL	
9 CYCL CALL	; Cycle call: pilot drilling
10 L Z+100 R0 FMAX	; Retract the tool
11 TOOL CALL 6 Z S3000	; Tool call: roughing/finishing (D12)
12 CYCL DEF 22 ROUGH-OUT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+350 ;FEED RATE F. ROUGHNG ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q19=+150 ;FEED RATE FOR RECIP. ~	

Q208=+99999 ;RETRACTION FEED RATE ~	
Q401=+100 ;FEED RATE FACTOR ~	
Q404=+0 ;FINE ROUGH STRATEGY	
13 CYCL CALL	; Cycle call: rough-out
14 CYCL DEF 23 FLOOR FINISHING ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+200 ;FEED RATE F. ROUGHNG ~	
Q208=+99999 ;RETRACTION FEED RATE	
15 CYCL CALL	; Cycle call: floor finishing
16 CYCL DEF 24 SIDE FINISHING ~	
Q9=+1 ;ROTATIONAL DIRECTION ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+400 ;FEED RATE F. ROUGHNG ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=-1 ;ROUGH-OUT TOOL	
17 CYCL CALL	; Cycle call: side finishing
18 L Z+100 R0 FMAX	; Retract the tool
19 M30	; End of program
20 LBL 1	; Contour subprogram 1: left pocket
21 CC X+35 Y+50	
22 L X+10 Y+50 RR	
23 C X+10 DR-	
24 LBL 0	
25 LBL 2	; Contour subprogram 2: right pocket
26 CC X+65 Y+50	
27 L X+90 Y+50 RR	
28 C X+90 DR-	
29 LBL 0	
30 LBL 3	; Contour subprogram 3: left square island
31 L X+27 Y+50 RL	
32 L Y+58	
33 L X+43	
34 L Y+42	
35 L X+27	
36 LBL 0	
37 LBL 4	; Contour subprogram 4: right triangular island
38 L X+65 Y+42 RL	
39 L X+57	
40 L X+65 Y+58	
41 L X+73 Y+42	
42 LBL 0	
43 END PGM 2 MM	

Example: Contour train



0 BEGIN PGM 3 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 10 Z S2000	; Tool call (diameter: 20)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 25 CONTOUR TRAIN ~	
Q1=-20 ;MILLING DEPTH ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q5=+0 ;SURFACE COORDINATE ~	
Q7=+250 ;CLEARANCE HEIGHT ~	
Q10=-5 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+200 ;FEED RATE F. ROUGHNG ~	
Q15=+1 ;CLIMB OR UP-CUT ~	
Q18=+0 ;COARSE ROUGHING TOOL ~	
Q446=+0.01 ;RESIDUAL MATERIAL ~	
Q447=+10 ;CONNECTION DISTANCE ~	
Q448=+2 ;PATH EXTENSION	
8 CYCL CALL	; Cycle call
9 L Z+250 R0 FMAX	; Retract the tool, end program
10 M30	
11 LBL 1	; Contour subprogram
12 L X+0 Y+15 RL	
13 L X+5 Y+20	
13 CT X+5 Y+75	
14 CT X+5 Y+75	
15 L Y+95	
16 RND R7.5	
17 L X+50	

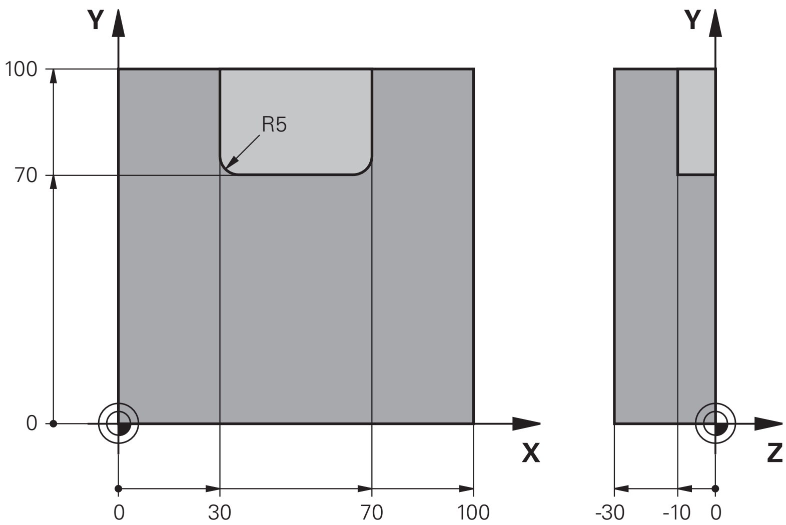
18 RND R7.5	
19 L X+100 Y+80	
20 LBL 0	
21 END PGM 3 MM	

Example: Open pocket and fine roughing with OCM cycles

The following NC program illustrates the use of OCM cycles. You will program an open pocket that is defined by means of an island and a boundary. Machining includes roughing and finishing of an open pocket.

Program sequence

- Tool call: Roughing cutter (Ø 20 mm)
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: Roughing cutter (Ø 8 mm)
- Define and call Cycle **272**
- Tool call: Finishing cutter (Ø 6 mm)
- Define and call Cycle **273**
- Define and call Cycle **274**



0 BEGIN PGM OCM_POCKET MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 10 Z S8000 F1500	; Tool call (diameter: 20 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF ~	
P1 = LBL 1 I2 = LBL 2	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-10 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR ~	
Q569=+1 ;OPEN BOUNDARY	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+10 ;PLUNGING DEPTH ~	

Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253= AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE	
Q438=+0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+6500 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+0 ;INFEED STRATEGY	
8 CYCL CALL	; Cycle call
9 TOOL CALL 4 Z S8000 F1500	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 272 OCM ROUGHING ~	
Q202=+10 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6000 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253= AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+10 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+0 ;INFEED STRATEGY	
12 CYCL CALL	; Cycle call
13 TOOL CALL 23 Z S10000 F2000	; Tool call (diameter: 6 mm)
14 L Z+100 R0 FMAX M3	
15 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+0.8 ;TOOL PATH OVERLAP ~	
Q385= AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
16 CYCL CALL	; Cycle call
17 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0 ;INFEED FOR FINISHING ~	
Q385= AUTO ;FINISHING FEED RATE ~	

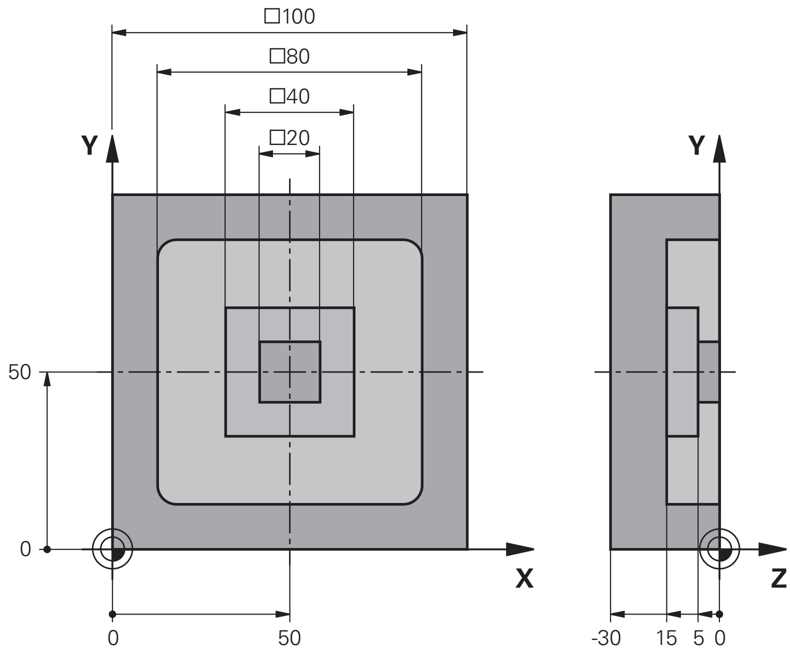
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB OR UP-CUT	
18 CYCL CALL	; Cycle call
19 M30	; End of program
20 LBL 1	; Contour subprogram 1
21 L X+0 Y+0	
22 L X+100	
23 L Y+100	
24 L X+0	
25 L Y+0	
26 LBL 0	
27 LBL 2	; Contour subprogram 2
28 L X+0 Y+0	
29 L X+100	
30 L Y+100	
31 L X+70	
32 L Y+70	
33 RND R5	
34 L X+30	
35 RND R5	
36 L Y+100	
37 L X+0	
38 L Y+0	
39 LBL 0	
40 END PGM OCM_POCKET MM	

Example: Program various depths with OCM cycles

The following NC program illustrates the use of OCM cycles. You will define one pocket and two islands at different heights. Machining includes roughing and finishing of a contour.

Program sequence

- Tool call: Roughing cutter (Ø 10 mm)
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: Finishing cutter (Ø 6 mm)
- Define and call Cycle **273**
- Define and call Cycle **274**



0 BEGIN PGM OCM_DEPTH MM	
1 BLK FORM 0.1 Z X-50 Y-50 Z-30	
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 TOOL CALL 5 Z S8000 F1500	; Tool call (diameter: 10 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF ~	
P1 = LBL 1 I2 = LBL 2 I3 = LBL 3 DEPTH5	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-15 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR ~	
Q569=+0 ;OPEN BOUNDARY	

7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253= AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
8 CYCL CALL	; Cycle call
9 TOOL CALL 23 Z S10000 F2000	; Tool call (diameter: 6 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+0.8 ;TOOL PATH OVERLAP ~	
Q385= AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
12 CYCL CALL	; Cycle call
13 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+0 ;INFEED FOR FINISHING ~	
Q385= AUTO ;FINISHING FEED RATE ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=+5 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB OR UP-CUT	
14 CYCL CALL	; Cycle call
15 M30	; End of program
16 LBL 1	; Contour subprogram 1
17 L X-40 Y-40	
18 L X+40	
19 L Y+40	
20 L X-40	
21 L Y-40	
22 LBL 0	

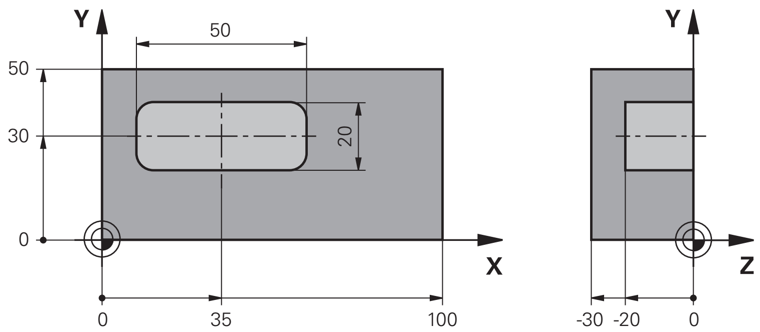
23 LBL 2	; Contour subprogram 2
24 L X-10 Y-10	
25 L X+10	
26 L Y+10	
27 L X-10	
28 L Y-10	
29 LBL 0	
30 LBL 3	; Contour subprogram 3
31 L X-20 Y-20	
32 L Y+20	
33 L X+20	
34 L Y-20	
35 L X-20	
36 LBL 0	
37 END PGM OCM_DEPTH MM	

Example: Face milling and fine roughing with OCM cycles

The following NC program illustrates the use of OCM cycles. You will face-mill a surface which will be defined by means of a boundary and an island. In addition, you will mill a pocket that contains an allowance for a smaller roughing tool.

Program sequence

- Tool call: Roughing cutter (Ø 12 mm)
- Program **CONTOUR DEF**
- Define Cycle **271**
- Define and call Cycle **272**
- Tool call: Roughing cutter (Ø 8 mm)
- Define Cycle **272** and call it again



0 BEGIN PGM FACE_MILL MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+50 Z+2	
3 TOOL CALL 6 Z S5000 F3000	; Tool call (diameter: 12 mm)
4 L Z+100 R0 FMAX M3	
5 CONTOUR DEF ~	
P1 = LBL 1 I2 = LBL 1 DEPTH2 ~	
P3 = LBL 2 ;	
6 CYCL DEF 271 OCM CONTOUR DATA ~	
Q203=+2 ;SURFACE COORDINATE ~	
Q201=-22 ;DEPTH ~	
Q368=+0 ;ALLOWANCE FOR SIDE ~	
Q369=+0 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR ~	
Q569=+1 ;OPEN BOUNDARY	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+24 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207=+8000 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253= AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=-1 ;ROUGH-OUT TOOL ~	

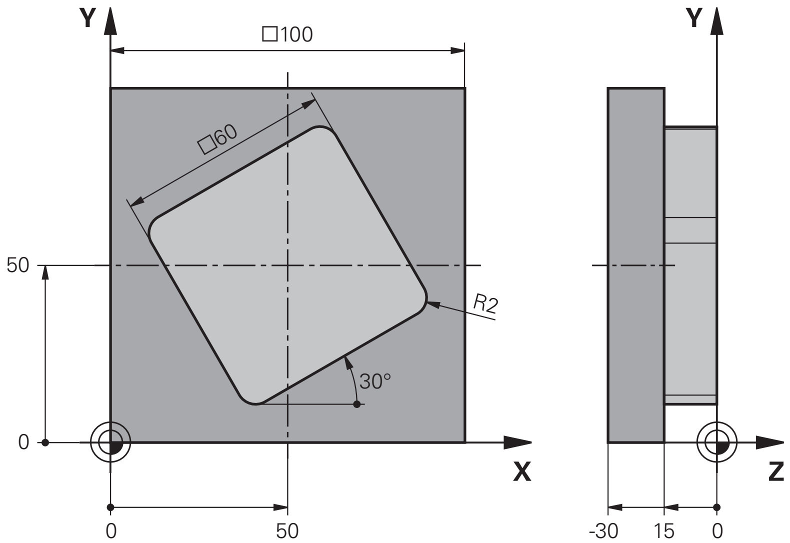
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+8000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
8 L X+0 Y+0 R0 FMAX M99	; Cycle call
9 TOOL CALL 4 Z S6000 F4000	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 272 OCM ROUGHING ~	
Q202=+25 ;PLUNGING DEPTH ~	
Q370=+0.4 ;TOOL PATH OVERLAP ~	
Q207= 6500 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253= AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+6 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
12 L X+0 Y+0 R0 FMAX M99	; Cycle call
13 M30	; End of program
14 LBL 1	; Contour subprogram 1
15 L X+0 Y+0	
16 L Y+50	
17 L X+100	
18 L Y+0	
19 L X+0	
20 LBL 0	
21 LBL 2	; Contour subprogram 2
22 L X+10 Y+30	
23 L Y+40	
24 RND R5	
25 L X+60	
26 RND R5	
27 L Y+20	
28 RND R5	
29 L X+10	
30 RND R5	
31 L Y+30	
32 LBL 0	
33 END PGM FACE_MILL MM	

Example: Contour with OCM figure cycles

The following NC program illustrates the use of OCM cycles. Machining includes roughing and finishing of a island.

Program sequence

- Tool call: Roughing cutter (Ø 8 mm)
- Define Cycle **1271**
- Define Cycle **1281**
- Define and call Cycle **272**
- Tool call: Finishing cutter (Ø 8 mm)
- Define and call Cycle **273**
- Define and call Cycle **274**

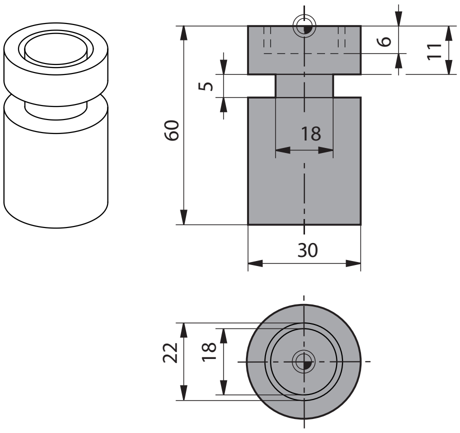


0 BEGIN PGM OCM_FIGURE MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 4 Z S8000 F1500	; Tool call (diameter: 8 mm)
4 L Z+100 R0 FMAX M3	
5 CYCL DEF 1271 OCM RECTANGLE ~	
Q650=+1 ;FIGURE TYPE ~	
Q218=+60 ;FIRST SIDE LENGTH ~	
Q219=+60 ;2ND SIDE LENGTH ~	
Q660=+0 ;CORNER TYPE ~	
Q220=+2 ;CORNER RADIUS ~	
Q367=+0 ;POCKET POSITION ~	
Q224=+30 ;ANGLE OF ROTATION ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-10 ;DEPTH ~	
Q368=+0.5 ;ALLOWANCE FOR SIDE ~	
Q369=+0.5 ;ALLOWANCE FOR FLOOR ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q578=+0.2 ;INSIDE CORNER FACTOR	

6 CYCL DEF 1281 OCM RECTANGLE BOUNDARY ~	
Q651=+100 ;LENGTH 1 ~	
Q652=+100 ;LENGTH 2 ~	
Q654=+0 ;POSITION REFERENCE ~	
Q655=+0 ;SHIFT 1 ~	
Q656=+0 ;SHIFT 2	
7 CYCL DEF 272 OCM ROUGHING ~	
Q202=+20 ;PLUNGING DEPTH ~	
Q370=+0.424 ;TOOL PATH OVERLAP ~	
Q207=+6800 ;FEED RATE MILLING ~	
Q568=+0.6 ;PLUNGING FACTOR ~	
Q253= AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+0 ;ROUGH-OUT TOOL ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q576=+10000 ;SPINDLE SPEED ~	
Q579=+0.7 ;PLUNGING FACTOR S ~	
Q575=+1 ;INFEED STRATEGY	
8 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
9 TOOL CALL 24 Z S10000 F2000	; Tool call (diameter: 8 mm)
10 L Z+100 R0 FMAX M3	
11 CYCL DEF 273 OCM FINISHING FLOOR ~	
Q370=+0.8 ;TOOL PATH OVERLAP	
Q385= AUTO ;FINISHING FEED RATE ~	
Q568=+0.3 ;PLUNGING FACTOR ~	
Q253= AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q438=+4 ;ROUGH-OUT TOOL ~	
Q595=+1 ;STRATEGY ~	
Q577=+0.2 ;APPROACH RADIUS FACTOR	
12 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
13 CYCL DEF 274 OCM FINISHING SIDE ~	
Q338=+15 ;INFEED FOR FINISHING ~	
Q385= AUTO ;FINISHING FEED RATE ~	
Q253= AUTO ;F PRE-POSITIONING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q438=+4 ;ROUGH-OUT TOOL ~	
Q351=+1 ;CLIMB OR UP-CUT	
14 L X+50 Y+50 R0 FMAX M99	; Positioning and cycle call
15 M30	; End of program
16 END PGM OCM_FIGURE MM	

Example: Interpolation turning with Cycle 291

The following NC program illustrates the use of Cycle **291 COUPLG.TURNG.INTERP.** This programming example shows how to machine an axial recess and a radial recess.



Tools

- Turning tool as defined in toolturn.trn: Tool no. 10: TO:1, ORI:0, TYPE:ROUGH, tool for axial recesses
- Turning tool as defined in toolturn.trn: Tool no. 11: TO:8, ORI:0, TYPE:ROUGH, tool for radial recesses

Program sequence

- Tool call: Tool for axial recess
- Start of interpolation turning: Description and call of Cycle **291**; **Q560** = 1
- End of interpolation turning: Description and call of Cycle **291**; **Q560** = 0
- Tool call: Recessing tool for radial recess
- Start of interpolation turning: Description and call of Cycle **291**; **Q560** = 1
- End of interpolation turning: Description and call of Cycle **291**; **Q560** = 0

> By converting parameter **Q561**, the turning tool is displayed in the simulation graphic as a milling tool.

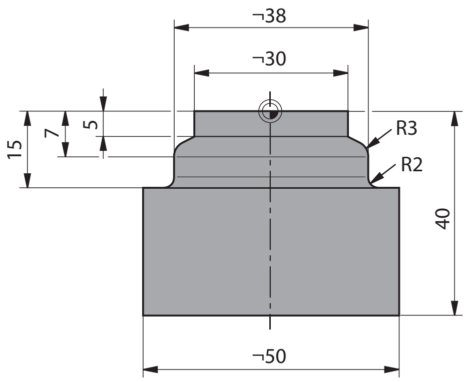
0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R15 L60	
2 TOOL CALL 10	; Tool call: tool for axial recess
3 CC X+0 Y+0	
4 LP PR+30 PA+0 R0 FMAX	; Retract the tool
5 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+1 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+1 ;CONVERT FROM TURNING TOOL	
6 CYCL CALL	; Call the cycle
7 LP PR+9 PA+0 RR FMAX	; Position the tool in the working plane
8 L Z+10 FMAX	
9 L Z+0.2 F2000	; Position the tool in the spindle axis

10 LBL 1	; Recessing on level surface (infeed: 0.2 mm, depth: 6 mm)
11 CP IPA+360 IZ-0.2 DR+ F10000	
12 CALL LBL 1 REP30	
13 LBL 2	; Retract from recess (step: 0.4 mm)
14 CP IPA+360 IZ+0.4 DR+	
15 CALL LBL 2 REP15	
16 L Z+200 R0 FMAX	; Retract to clearance height, deactivate radius compensation
17 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+0 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+0 ;CONVERT FROM TURNING TOOL	
18 CYCL CALL	; Call the cycle
19 TOOL CALL 11	; Tool call: tool for radial recess
20 CC X+0 Y+0	
21 LP PR+25 PA+0 R0 FMAX	; Retract the tool
22 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+1 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+1 ;CONVERT FROM TURNING TOOL	
23 CYCL CALL	; Call the cycle
24 LP PR+15 PA+0 RR FMAX	; Position the tool in the working plane
25 L Z+10 FMAX	
26 L Z-11 F7000	; Position the tool in the spindle axis
27 LBL 3	; Recessing on lateral surface (infeed: 0.2 mm, depth: 6 mm)
28 CC X+0.1 Y+0	
29 CP IPA+180 DR+ F10000	
30 CC X-0.1 Y+0	
31 CP IPA+180 DR+	
32 CALL LBL 3 REP15	
33 LBL 4	; Retract from recess (step: 0.4 mm)
34 CC X-0.2 Y+0	
35 CP PA+180 DR+	
36 CC X+0.2 Y+0	
37 CP IPA+180 DR+	
38 CALL LBL 4 REP8	
39 LP PR+50 FMAX	

40 L Z+200 R0 FMAX	; Retract to clearance height, deactivate radius compensation
41 CYCL DEF 291 COUPLG.TURNG.INTERP. ~	
Q560=+0 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q216=+0 ;CENTER IN 1ST AXIS ~	
Q217=+0 ;CENTER IN 2ND AXIS ~	
Q561=+0 ;CONVERT FROM TURNING TOOL	
42 CYCL CALL	; Call the cycle
43 TOOL CALL 11	; Repeated **TOOL CALL** in order to reset the conversion of parameter Q561
44 M30	
45 END PGM 5 MM	

Example: Interpolation Turning Cycle 292

The following NC program illustrates the use of Cycle **292 CONTOUR.TURNG.INTRP.** This programming example shows how to machine an outside contour with the milling spindle rotating.



Program sequence

- Tool call: Milling cutter D20
- Cycle **32 TOLERANCE**
- Reference to the contour with Cycle **14**
- Cycle **292 CONTOUR.TURNG.INTRP.**

0 BEGIN PGM 6 MM	
1 BLK FORM CYLINDER Z R25 L40	
2 TOOL CALL 10 Z S111	; Tool call: end mill D20
*** - ...**	; Use Cycle 32 to define the tolerance
3 CYCL DEF 32.0 TOLERANZ	
4 CYCL DEF 32.1 T0.05	
5 CYCL DEF 32.2 HSC-MODE:1	
6 CYCL DEF 14.0 CONTOUR	
7 CYCL DEF 14.1 CONTOUR LABEL1	
8 CYCL DEF 292 CONTOUR.TURNG.INTRP. ~	
Q560=+1 ;SPINDLE COUPLING ~	
Q336=+0 ;ANGLE OF SPINDLE ~	
Q546=+3 ;CHANGE TOOL DIRECTN. ~	
Q529=+0 ;MACHINING OPERATION ~	
Q221=+0 ;SURFACE OVERSIZE ~	
Q441=+1 ;INFEED ~	
Q449=+15000 ;FEED RATE ~	
Q491=+15 ;CONTOUR START RADIUS ~	
Q357=+2 ;CLEARANCE TO SIDE ~	
Q445=+50 ;CLEARANCE HEIGHT	
9 L Z+50 R0 FMAX M3	; Pre-position in the tool axis, spindle ON
10 L X+0 Y+0 R0 FMAX M99	; Pre-position in the working plane to the center of rotation, call the cycle
11 M30	; End of program
12 LBL 1	; LBL1 contains the contour

13 L Z+2 X+15	
14 L Z-5	
15 L Z-7 X+19	
16 RND R3	
17 L Z-15	
18 RND R2	
19 L X+27	
20 LBL 0	
21 END PGM 6 MM	

14.4 Cycles for milling and turning

14.4.1 Overview

The control offers the following cycles for turning operations:

Special cycles

Cycle		Call	Further information
800	**ADJUST XZ SYSTEM** (option 50) ■ Moving the tool to a suitable position relative to the turning spindle	**DEF**-active	Page 681
801	**RESET ROTARY COORDINATE SYSTEM** (option 50) ■ Resetting of cycle **800**	**DEF**-active	Page 689
892	**CHECK UNBALANCE** (option 50) ■ Checking the unbalance of the turning spindle	**DEF**-active	Page 690

Longitudinal turning cycles

Cycle		Call	Further information
811	**SHOULDER, LONGITDNL.** (option 50) ■ Longitudinal turning of rectangular shoulders	**CALL**-active	Page 694
812	**SHOULDER, LONG. EXT.** (option 50) ■ Longitudinal turning of rectangular shoulders ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 698
813	**TURN PLUNGE CONTOUR LONGITUDINAL** (option 50) ■ Longitudinal turning of shoulders with plunging elements	**CALL**-active	Page 703
814	**TURN PLUNGE LONGITUDINAL EXT.** (option 50) ■ Longitudinal turning of shoulders with plunging elements ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 707
810	**TURN CONTOUR LONG.** (option 50) ■ Longitudinal turning of turning contours of any shape ■ Removing stock paraxially	**CALL**-active	Page 712
815	**CONTOUR-PAR. TURNING** (option 50) ■ Longitudinal turning of turning contours of any shape ■ Removing of stock is performed parallel to the contour	**CALL**-active	Page 717

Face turning cycles

Cycle		Call	Further information
821	**SHOULDER, FACE** (option 50) ■ Face turning of rectangular shoulders	**CALL**-active	Page 721
822	**SHOULDER, FACE. EXT.** (option 50) ■ Face turning of rectangular shoulders ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 724
823	**TURN TRANSVERSE PLUNGE** (option 50) ■ Face turning of shoulders with plunging elements	**CALL**-active	Page 728
824	**TURN PLUNGE TRANSVERSE EXT.** (option 50) ■ Face turning of shoulders with plunging elements ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 732
820	**TURN CONTOUR TRANSV.** (option 50) ■ Face turning of turning contours of any shape	**CALL**-active	Page 736

Recess-turning cycles

Cycle		Call	Further information
841	**SIMPLE REC. TURNG., RADIAL DIR.** (option 50) ■ Recess turning of rectangular slots in longitudinal direction	**CALL**-active	Page 742
842	**ENH.REC.TURNNG, RAD.** (option 50) ■ Recess turning of slots in longitudinal direction ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 746
851	**SIMPLE REC TURNG, AX** (option 50) ■ Recess turning of slots in transverse direction	**CALL**-active	Page 751
852	**ENH.REC.TURNING, AX.** (option 50) ■ Recess turning of slots in transverse direction ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 755
840	**RECESS TURNG, RADIAL** (option 50) ■ Recess turning of slots of any shape in longitudinal direction	**CALL**-active	Page 760
850	**RECESS TURNG, AXIAL** (option 50)	**CALL**-active	Page 765

Cycle		Call	Further information
	■ Recess turning of slots of any shape in transverse direction ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface		

Recessing cycles

Cycle		Call	Further information
861	**SIMPLE RECESS, RADL.** (option 50) ■ Radial recessing of rectangular slots	**CALL**-active	Page 770
862	**EXPND. RECESS, RADL.** (option 50) ■ Radial recessing of rectangular slots ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 774
871	**SIMPLE RECESS, AXIAL** (option 50) ■ Axial recessing of rectangular slots	**CALL**-active	Page 779
872	**EXPND. RECESS, AXIAL** (option 50) ■ Axial recessing of rectangular slots ■ Rounding arcs at contour corners ■ Chamfer or rounding arc at the start and end of the contour ■ Angle for plane and circumferential surface	**CALL**-active	Page 785
860	**CONT. RECESS, RADIAL** (option 50) ■ Radial recessing of slots of any shape	**CALL**-active	Page 791
870	**CONT. RECESS, AXIAL** (option 50) ■ Axial recessing of slots of any shape	**CALL**-active	Page 796

Thread cutting cycles

Cycle		Call	Further information
831	**THREAD LONGITUDINAL** (option 50) ■ Longitudinal turning of threads	**CALL**-active	Page 801
832	**THREAD EXTENDED** (option 50) ■ Longitudinal or face turning of threads and tapered threads ■ Definition of an approach path and an idle travel path	**CALL**-active	Page 805
830	**THREAD CONTOUR-PARALLEL** (option 50) ■ Longitudinal or face turning of threads of any shape ■ Definition of an approach path and an idle travel path	**CALL**-active	Page 809

Extended turning cycles

Cycle		Call	Further information
882	**SIMULTANEOUS ROUGHING FOR TURNING** (option 50 & 158) ■ Roughing of complex contours with different angles of inclination	**CALL**-active	Page 816
883	**TURNING SIMULTANEOUS FINISHING** (option 50 & 158) ■ Finishing of complex contours with different angles of inclination	**CALL**-active	Page 821

14.4.2 Working with turning cycles

Working with turning cycles

In turning cycles, the control takes the cutting geometry (**TO, RS, P-ANGLE, T-ANGLE**) of the tool into account in order to prevent damage to the defined contour elements. If it is not possible to machine the entire contour with the active tool, the control will display a warning.

You can use the turning cycles both for inside and outside machining. Depending upon the specific cycle, the control detects the machining position (inside or outside machining) via the starting position or tool position when the cycle is called. In some cycles you can also enter the machining position directly in the cycle. After modifying the machining position, check the tool position and the direction of rotation.

If you program **M136** before a cycle, the control interprets feed rate values in the cycle in mm/rev.; without **M136** in mm/min.

If you execute turning cycles with inclined machining (**M144**), the angles of the tool with respect to the contour change. The control automatically takes these modifications into account and thus also monitors the machining in inclined state to prevent contour damage.

Some cycles machine contours that you have written in a subprogram. You can program these contours with Klartext contouring functions. Before calling the cycle, you must program the cycle **14 CONTOUR** to define the subprogram number.

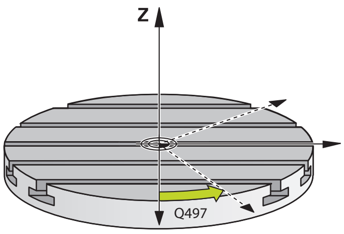
The turning cycles 81x - 87x as well 880, 882, and 883 must be called with **CYCL CALL** or **M99**. Before programming a cycle call, be sure to program:

- Turning mode: **FUNCTION MODE TURN**
- Call a tool with **TOOL CALL**
- Direction of rotation of turning spindle, e.g. **M303**
- Selection of speed or cutting speed: **FUNCTION TURNDATA SPIN**
- If you use feed rate per revolution mm/rev., **M136**
- Position the tool to a suitable starting point e.g. **L X+130 Y+0 R0 FMAX**
- Adapt the coordinate system, and align the tool: **CYCL DEF 800 ADJUST XZ SYSTEM**

14.4.3 Cycle 800 ADJUST XZ SYSTEM

Application

	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer. The cycle is machine-dependent.



To be able to perform a turning operation, you need to position the tool appropriately relative to the turning spindle. For this purpose, you can use Cycle **800 ADJUST XZ SYSTEM**.

With turning operations, the inclination angle between the tool and turning spindle is important, in order to e.g. machine contours with undercuts. Cycle **800** provides various possibilities for aligning the coordinate system for an inclined machining operation:

- If you have positioned the tilting axis for inclined machining, you can use Cycle **800** to orient the coordinate system to the positions of the tilting axes (**Q530=0**). In this case, make sure to program **M144** or **M128**/**TCPM** for proper calculation of the orientation
- Cycle **800** calculates the required tilting axis angle based on the inclination angle **Q531** – depending on the strategy selected in the **INCLINED MACHINING Q530** parameter, the control positions the tilting axis with (**Q530=1**) or without compensating movement (**Q530=2**)
- Cycle **800** uses the inclination angle **Q531** to calculate the required tilting axis angle, but does not position the tilting axis (**Q530=3**). You need to position the tilting axis manually to the calculated values **Q120** (A axis), **Q121** (B axis), and **Q122** (C axis) after the cycle

If the milling spindle axis and the turning spindle axis are parallel to each other, you can use the **Precession angle Q497** to define any desired rotation of the coordinate system about the spindle axis (Z axis). This may be necessary if you have to bring the tool into a specific position due to space restrictions or if you want to improve your ability to observe a machining process. If the turning spindle and milling spindle axes are not parallel, only two precession angles are realistic for machining. The control selects the angle that is closest to the input value of **Q497**.

Cycle **800** positions the milling spindle such that the cutting edge is aligned relative to the turning contour. You can use a mirrored version of the tool (**REVERSE TOOL Q498**); this offsets the milling spindle by 180°. In this way, you can use your tools for both inside and outside machining. Position the cutting edge at the center of the turning spindle using a positioning block, such as **L Y+0 R0 FMAX**.

- If you change the position of a tilting axis, you need to run Cycle **800** again to align the coordinate system.
- Check the orientation of the tool before machining.

Eccentric turning

Sometimes it is not possible to clamp a workpiece such that the axis of rotation is aligned with the axis of the turning spindle. For example, this is the case with large or rotationally non-symmetric workpieces. The eccentric turning **Q535** function in Cycle **800** enables you to perform turning in such cases as well.

During eccentric turning, more than one linear axis is coupled to the turning spindle. The control compensates the eccentricity by performing circular compensating movements with the coupled linear axes.

This function must be enabled and adapted by the machine manufacturer.

If you machine with high speed and a high amount of eccentricity, you need to program large feed rates for the linear axes in order to perform the movements synchronously. If these feed rates are not met, the contour would be damaged. The control therefore generates an error message if 80 % of a maximum axis speed or acceleration is exceeded. If this occurs, reduce the speed.

Operating information

NOTICE

Danger of collision!

The control performs compensating movements during coupling and decoupling. Check for possible collisions.

- Coupling and decoupling must be performed while the spindle is stationary.

NOTICE

Danger of collision!

Collision monitoring (DCM) is not active during eccentric turning. The control displays a corresponding warning during eccentric turning.

- Watch out for possible collisions!

NOTICE

Danger of collision!

The rotation of the workpiece creates centrifugal forces that lead to vibration (resonance), depending on the unbalance. This vibration has a negative effect on the machining process and reduces the tool life.

- Select the technology data in such a way that no vibrations (resonances) occur.

- Turn a test cut before the actual machining operation to ensure that the required speeds can be attained.
- The linear axis positions resulting from the compensation are displayed by the control only in the ACTUAL value position display.

Effect

With Cycle **800 ADJUST XZ SYSTEM**, the control aligns the workpiece coordinate system and orients the tool correspondingly. Cycle **800** is effective until it is reset by Cycle **801**, or until Cycle **800** is redefined. Some cycle functions of Cycle **800** are implicitly reset by other factors:

- Mirroring of tool data (**Q498 REVERSE TOOL**) is reset by a tool call with **TOOL CALL**
- The **ECCENTRIC TURNING Q535** function is reset at the end of the program or if the program is aborted (internal stop)

Notes

The machine tool builder configures your machine tool. If the tool spindle was defined as an axis in the kinematic model during this configuration, the feed-rate potentiometer is effective for movements related to Cycle **800**.

The machine manufacturer can configure a grid for the positioning of the tool spindle.

NOTICE

Danger of collision!

If the milling spindle was defined as an NC axis in turning mode, then the control is able to derive a tool reversal from the axis position. However, if the milling spindle was defined as a spindle, there is a risk that the tool reversal definition might get lost!
In either case, proceed as follows:

- Enable tool reversal again after a **TOOL CALL** block

NOTICE

Danger of collision!

If **Q498**=1 and you additionally program the **FUNCTION LIFTOFF ANGLE TCS** function, then there might be different results, depending on the configuration. If the tool spindle has been defined as an axis, the **LIFTOFF** will be included in the rotation during tool reversal. If the tool spindle has been defined as a kinematic transformation, then the **LIFTOFF** will **not** be included in the rotation during tool reversal!

- Carefully test the NC program or program section in the **Program Run** mode **Single Block**
- If required, change the algebraic sign of the SPB angle.

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool must be clamped and measured in the correct position.
- Cycle **800** positions only the first rotary axis based on the tool position. If a **M138** is activated, then this limits the selection to the defined rotary axis. If you want to move other rotary axes to a specific position, position these axis correspondingly before running Cycle **800**.

  Further information: "Taking rotary axes into account during machining operations with M138", Page 1251

Notes on programming

- You can mirror the tool data (**Q498 REVERSE TOOL**) only if a turning tool has been selected.
- To reset Cycle **800**, program Cycle **801 RESET ROTARY COORDINATE SYSTEM**.
- Cycle **800** limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine tool builder) and the amount of eccentricity. You might have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle **800**. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle **800**, the smaller value will be applied. To reset Cycle **800**, program Cycle **801**. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** takes effect again.
- If the workpiece is to be rotated about the workpiece spindle, then use an offset of the workpiece spindle in the preset table. Basic rotations are not permitted; the control issues an error message.
- If, in parameter **Q530** “Inclined machining,” you use the 0 setting (tilting axes must be positioned beforehand), then you must program an **M144** or **TCPM/M128** ahead of time.
- If, in parameter **Q530** “Inclined machining,” you use the settings 1: MOVE, 2: TURN and 3: STAY, then the control, depending on the machine configuration, activates function **M144** or TCPM

  Further information: "Turning (option 50)", Page 212

Cycle parameters

Help graphic	Parameters
	Q497 Precession angle? Angle at which the control positions the tool. Input: **0.0000...359.9999**
	Q498 Reverse tool (0=no/1=yes)? Mirror tool for inside/outside machining. Input: **0**, **1**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **0**: Maintain tilting axis position (axis must be positioned beforehand) **1**: Automatically position the tilting axis, and orient the tool tip (MOVE). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (TURN) **3**: Do not position the tilting axis. Position the tilting axes later in a separate positioning block (STAY). The control stores the position values in the parameters **Q120** (A axis), **Q121** (B axis) and **Q122** (C axis). Input: **0**, **1**, **2**, **3**
	Q531 Angle of incidence? Angle of incidence for positioning the tool Input: **-180...+180**
	Q532 Feed rate for positioning? Traversing speed of the tilting axis during automatic positioning Input: **0.001...99999.999**, or **FMAX**
	Q533 Preferred dir. of incid. angle? **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**

Help graphic	Parameters
	Q535 Eccentric turning? Couple the axes for the eccentric turning operation: **0**: Deactivate axis couplings **1**: Activate axis couplings. The center of rotation is located at the active preset **2**: Activate axis couplings. The center of rotation is located at the active datum **3**: Do not change the axis couplings Input: **0**, **1**, **2**, **3**
	Q536 Eccentric turning without stop? Interrupt program run before the axes are coupled: **0**: Stop before the axes are coupled again. In stopped condition, the control opens a window in which the amount of eccentricity and the maximum deflection of the individual axes are displayed. You can then continue the machining operating with **NC-Start** or select **ABBRUCH** **1**: Axes are coupled without stopping beforehand Input: **0**, **1**
	Q599 or **QS599 Retraction path/macro?** Retraction prior to execution of positioning in the rotary axis or tool axis: **0**: No retraction **-1**: Maximum retraction with **M140 MB MAX**, see "Retracting in the tool axis with M140", Page 1252 **>0**: Path for the retraction in **mm** or **inch** **"..."**: Path for an NC program that is to be called as a user macro. **Further information:** "User macro", Page 688 Input: **-1...9999** in the case of text entry maximum **255** characters, alternatively **QS** parameter

Example

```
11 CYCL DEF 800 ADJUST XZ SYSTEM ~
    Q497=+0                 ;PRECESSION ANGLE ~
    Q498=+0                 ;REVERSE TOOL ~
    Q530=+0                 ;INCLINED MACHINING ~
    Q531=+0                 ;ANGLE OF INCIDENCE ~
    Q532=+750               ;FEED RATE ~
    Q533=+0                 ;PREFERRED DIRECTION ~
    Q535=+3                 ;ECCENTRIC TURNING ~
    Q536=+0                 ;ECCENTRIC W/O STOP ~
    Q599=-1                 ;RETRACT
```

User macro

The user macro is another NC program.

A user macro contains a sequence of multiple instructions. With a macro, you can define multiple NC functions that the control executes. As a user, you create macros as an NC program.

Macros work in the same manner as NC programs that are called with the **PGM CALL** function, for example. You define a macro as an NC program with the file type *.h.

- HEIDENHAIN recommends using QL parameters in the macro. QL parameters have only a local effect for an NC program. If you use other types of variables in the macro, then changes may also have an effect on the calling NC program. In order to explicitly cause changes in the calling NC program, use Q or QS parameters with the numbers 1200 to 1399.
- Within the macro, you can read the value of the cycle parameters.

  Further information: "Variables: Q, QL, QR and QS parameters", Page 1268

Example of a user macro for retraction

0 BEGIN PGM RET MM	
1 FUNCTION RESET TCPM	; reset TCPM
2 L Z-1 R0 FMAX M91	; Traverse with M91
3 FN 10: IF +Q533 NE +0 GOTO LBL "DEF_DIRECTION"	; If Q533 (preferred direction from Cycle 800) is not equal to 0, then jump to LBL "DEF_DIRECTION"
4 FN 18: SYSREAD QL1 = ID240 NR1 IDX4	; Read system data (nominal position in the REF system) and store in QL1
5 QL0 = 500 * SGN QL1	; SGN = Check algebraic sign
6 FN 9: IF +0 EQU +0 GOTO LBL "MOVE"	; Jump to LBL MOVE
7 LBL "DIRECTION"	
8 QL0 = 500 * SGN Q533	; SGN = Check algebraic sign
9 LBL "MOVE"	
10 L X-500 Y+QL0 R0 FMAX M91	; Retraction with M91
11 END PGM RET MM	

14.4.4 Cycle 801 RESET ROTARY COORDINATE SYSTEM

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
> The cycle is machine-dependent.

Cycle **801** resets the following settings you have programmed with Cycle **800**:

- Precession angle **Q497**
- Reverse tool **Q498**

If you executed the eccentric turning function with Cycle **800**, please note the following: Cycle **800** limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine tool builder) and the amount of eccentricity. You might have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle **800**. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle **800**, the smaller value will be applied. To reset Cycle **800**, program Cycle **801**. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** takes effect again.

> Cycle **801** does not orient the tool to the starting position. If a tool was oriented with Cycle **800**, it remains in this position also after resetting.

Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- With Cycle **801 RESET ROTARY COORDINATE SYSTEM**, you can reset the settings you have made with Cycle **800 ADJUST XZ SYSTEM**.

Notes on programming

- To reset Cycle **800**, program Cycle **801 RESET ROTARY COORDINATE SYSTEM**.
- Cycle **800** limits the maximum spindle speed permitted for eccentric turning. It results from a machine-dependent configuration (defined by your machine tool builder) and the amount of eccentricity. You might have programmed a speed limitation with **FUNCTION TURNDATA SMAX** before programming Cycle **800**. If the value of this speed limitation is smaller than the speed limitation calculated by Cycle **800**, the smaller value will be applied. To reset Cycle **800**, program Cycle **801**. This will also reset the speed limitation set by that cycle. After that, the speed limitation programmed before the cycle call with **FUNCTION TURNDATA SMAX** takes effect again.

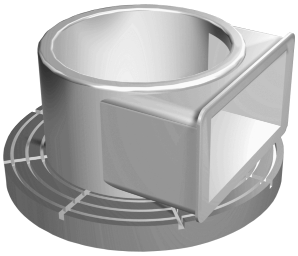
Cycle parameters

Help graphic	Parameter
	Cycle **801** does not have a cycle parameter. Close cycle input with the **END** key.

14.4.5 Cycle 892 CHECK UNBALANCE

Application

	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



An unbalance can occur when turning an unsymmetrical workpiece, such as a pump body. This may cause a high load on the machine, depending on the rotational speed, mass and shape of the workpiece. With Cycle **892 CHECK UNBALANCE**, the control checks the unbalance of the turning spindle. This cycle uses two parameters. **Q450** describes the maximum unbalance and **Q451** the maximum spindle speed. **If the maximum unbalance is exceeded, an error message is displayed and the NC program is aborted.** If the maximum unbalance is not exceeded, the control executes the NC program without interruption. This function protects the machine mechanics. It enables you to take action if an excessive unbalance is detected.

Notes

Your machine tool builder configures Cycle **892**.

Your machine tool builder defines the function of Cycle **892**.

The turning spindle rotates during the unbalance check.

This function can also be run on machines with more than one turning spindle. Contact the machine tool builder for further information.

You need to check the applicability of the control's internal unbalance functionality for each of your machine types. If the unbalance amplitude of the turning spindle has very little effect on the adjoining axes, it might not be possible to calculate useful unbalance values from the determined results. In this case, you will have to use a system with external sensors for unbalance monitoring.

NOTICE

Danger of collision!

Check the unbalance whenever you clamp a new workpiece. If required, use balancing weights to compensate any imbalance. If high unbalance loads are not compensated for, then this may lead to defects on the machine.

- Before starting a new machining cycle, run Cycle **892**.
- If required, use balancing weights to compensate for any unbalance.

NOTICE

Danger of collision!

The removal of material during machining will change the mass distribution within the workpiece. This generates the unbalance, which is why an unbalance test is recommended even between the machining steps. If high unbalance loads are not compensated, then this may lead to defects on the machine

- Make sure to also run Cycle **892** between the machining steps.
- If required, use balancing weights to compensate for any unbalance.

NOTICE

Danger of collision!

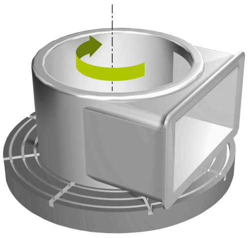
High unbalance loads, especially in combination with a high mass, may damage the machine. Consider the mass and unbalance of the workpiece when choosing the speed.

- Do not program high speeds with heavy workpieces or high unbalance loads.

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- If Cycle **892 CHECK UNBALANCE** has aborted the NC program, then we recommend that you use the manual MEASURE UNBALANCE cycle. With this cycle, the control determines the unbalance and calculates the mass and position of a balancing weight.

  Further information: "Unbalance in turning operations", Page 222

Cycle parameters

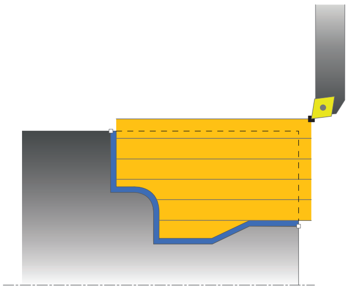
Help graphic	Parameter
	**Q450 Max. permissible runout?** Specifies the maximum runout of a sinusoidal unbalance signal in millimeters (mm). The signal results from the following error of the measuring axis and from the spindle revolutions. Input: **0...99999.9999**
	Q451 Rotational speed? Enter the rotational speed in revolutions per minute. The test for an unbalance begins with a low initial speed (e.g., 50 rpm). It is then automatically increased by specified increments (e.g., 25 rpm). until the maximum speed defined in parameter **Q451** is reached. Spindle speed override is disabled. Input: **0...99999**

Example

```
11 CYCL DEF 892 CHECK UNBALANCE ~
    Q450=+0              ;MAXIMUM RUNOUT ~
    Q451=+50             ;SPEED
```

14.4.6 Fundamentals of turning cycles

> Refer to your machine manual.
> Machine and control must be specially prepared by the machine manufacturer for use of this cycle.
> Option 50 must have been enabled.



The pre-positioning of the tool has a decisive influence on the workspace of the cycle and thus the machining time. During roughing, the starting point for cycles corresponds to the tool position when the cycle is called. When calculating the area to be machined, the control takes into account the starting point and the end point defined in the cycle or of the contour defined in the cycle. If the starting point is within the area to be machined, then the control positions the tool at the set-up clearance beforehand in some cycles.

The direction of stock removal is longitudinal to the rotary axis for Cycles **81x** and transverse to the rotary axis for Cycles **82x**. In Cycle **815**, the movements are contour-parallel.

The cycles can be used for inside and outside machining. The control takes the information for this from the position of the tool or from the definition in the cycle.

Further information: "Working with turning cycles", Page 680

For cycles in which a defined contour is machined (Cycles **810**, **820**, and **815**), the direction set when programming the contour determines the machining direction.

In cycles for turning you can specify the machining strategies of roughing, finishing or complete machining.

NOTICE

Danger of collision!

The turning cycles position the tool automatically to the starting point during finishing. The approach strategy is influenced by the position of the tool when the cycle is called. The decisive factor is whether the tool is located inside or outside an envelope contour when the cycle is called. The envelope contour is the programmed contour, enlarged by the set-up clearance. If the tool is within the envelope contour, the cycle positions the tool at the defined feed rate directly to the starting position. This can cause contour damage.

- Position the tool at a sufficient distance from the starting point to prevent the possibility of contour damage
- If the tool is outside the envelope contour, positioning to the envelope contour is performed at rapid traverse, and at the programmed feed rate within the envelope contour.

The control monitors the length of the cutting edge **CUTLENGTH** in the turning cycles. If the cutting depth programmed in the turning cycle is greater than the length of the cutting edge defined in the tool table, then the control issues a warning. In this case, the cutting depth will be reduced automatically in the machining cycle.

Execution with a FreeTurn tool

The control supports the execution of the contours with FreeTurn tools in the cycles **81x** and **82x**. This method allows you to perform the most common turning operation with just one tool. Thanks to the flexible tool, machining times can be reduced because the control does not need to change tools as much.

Requirements

- The tool must be correctly defined.
  "Turning operation with FreeTurn tools"

NOTICE

Danger of collision!

The shaft length of the turning tool limits the diameter that can be machined. There is a risk of collision during machining!

- Check the machining sequence in the simulation

The NC program remains unchanged except for the calling of the FreeTurn cutting edge.

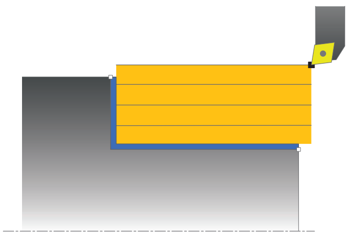
Further information: "Example: Turning with a FreeTurn tool", Page 833

14.4.7 Cycle 811 SHOULDER, LONGITDNL.

Application

Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to carry out longitudinal turning of right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run

The cycle processes the area from the tool position to the end point defined in the cycle.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2 The control performs a paraxial infeed movement at rapid traverse.
3 The control finishes the contour of the finished part at the defined feed rate **Q505**.
4 The control retracts the tool at the defined feed rate to the set-up clearance.
5 The control returns the tool at rapid traverse to the cycle starting point.

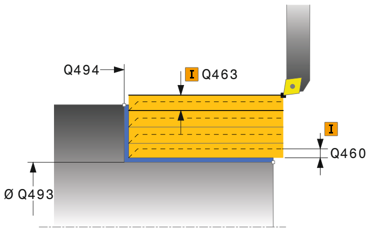
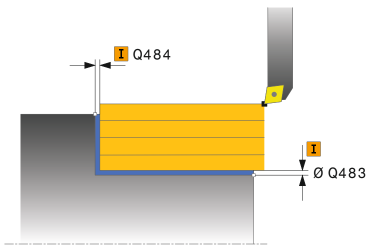
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
 	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
 	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

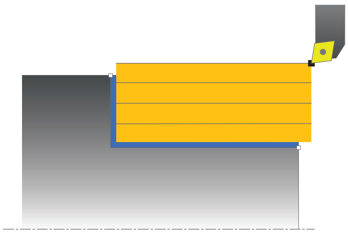
Example

```
11 CYCL DEF 821 SHOULDER, LONGITDNL. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-55                ;CONTOUR END IN Z ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q506=+0                 ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 R0 FMAX M303
13 CYCL CALL
```

14.4.8 Cycle 812 SHOULDER, LONG. EXT.

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run longitudinal turning of shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the X coordinate and then in the Z coordinate to set-up clearance and starts the cycle there.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

If the starting point lies in the area to be machined, the control positions the tool to set-up clearance beforehand.

1. The control performs a paraxial infeed movement at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

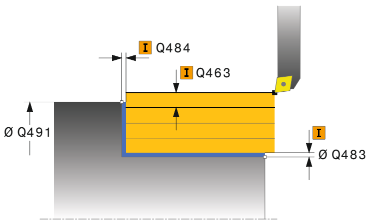
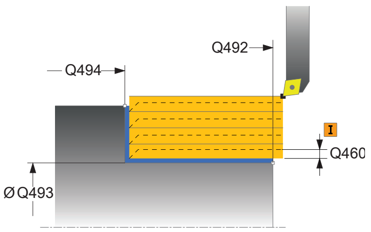
Notes

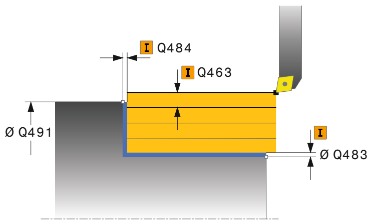
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	**Q491 Diameter at contour start?** X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of circumferen. surface? Angle between the circumferential surface and rotary axis Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of face? Angle between the plane surface and the rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

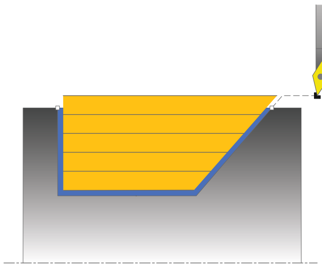
Example

```
11 CYCL DEF 812 SHOULDER, LONG. EXT. ~
    Q215=+0                   ;MACHINING OPERATION ~
    Q460=+2                   ;SAFETY CLEARANCE ~
    Q491=+75                  ;DIAMETER AT CONTOUR START ~
    Q492=+0                   ;CONTOUR START IN Z ~
    Q493=+50                  ;DIAMETER AT CONTOUR END ~
    Q494=-55                  ;CONTOUR END IN Z ~
    Q495=+5                   ;ANGLE OF CIRCUM. SURFACE ~
    Q501=+1                   ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5                 ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5                 ;RADIUS OF CONTOUR EDGE ~
    Q496=+0                   ;ANGLE OF FACE ~
    Q503=+1                   ;TYPE OF END ELEMENT ~
    Q504=+0.5                 ;SIZE OF END ELEMENT ~
    Q463=+3                   ;MAX. CUTTING DEPTH ~
    Q478=+0.3                 ;ROUGHING FEED RATE ~
    Q483=+0.4                 ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                 ;OVERSIZE IN Z ~
    Q505=+0.2                 ;FINISHING FEED RATE ~
    Q506=+0                   ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.9 Cycle 813 TURN PLUNGE CONTOUR LONGITUDINAL

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run longitudinal turning of shoulders with plunging elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1. The infeed movement is performed at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

Notes

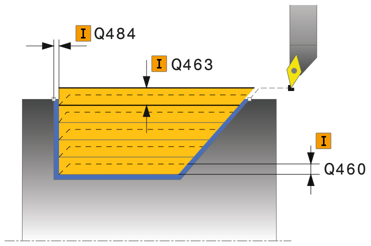
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

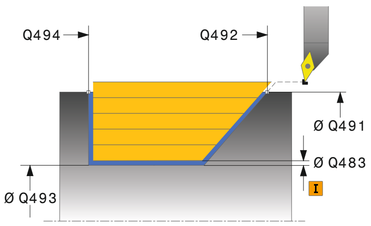
Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter





Q215 Machining operation (0/1/2/3)?
Define extent of machining:
0: Roughing and finishing
1: Only roughing
2: Only finishing to final dimension
3: Only finishing to oversize
Input: **0**, **1**, **2**, **3**

Q460 Set-up clearance?
Distance for retraction and prepositioning. This value has an incremental effect.
Input: **0...999.999**

Q491 Diameter at contour start?
X coordinate of the contour starting point (diameter value)
Input: **-99999.999...+99999.999**

Q492 Contour start in Z?
Z coordinate of the starting point for the plunging path
Input: **-99999.999...+99999.999**

Q493 Diameter at end of contour?
X coordinate of the contour end point (diameter value)
Input: **-99999.999...+99999.999**

Q494 Contour end in Z?
Z coordinate of the contour end point
Input: **-99999.999...+99999.999**

Q495 Angle of side?
Angle of plunging flank. The reference angle is the line perpendicular to the rotary axis.
Input: **0...89.9999**

Q463 Maximum cutting depth?
Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts.
Input: **0...99.999**

Q478 Roughing feed rate?
Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute.
Input: **0...99999.999** or **FAUTO**

Q483 Oversize for diameter?
Diameter oversize on the defined contour. This value has an incremental effect.
Input: **0...99.999**

Help graphic	Parameter
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

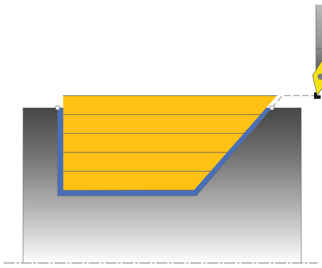
Example

```
11 CYCL DEF 813 TURN PLUNGE CONTOUR LONGITUDINAL ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q491=+75                 ;DIAMETER AT CONTOUR START ~
    Q492=-10                 ;CONTOUR START IN Z ~
    Q493=+50                 ;DIAMETER AT CONTOUR END ~
    Q494=-55                 ;CONTOUR END IN Z ~
    Q495=+70                 ;ANGLE OF SIDE ~
    Q463=+3                  ;MAX. CUTTING DEPTH ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q506=+0                  ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 R0 FMAX M303
13 CYCL CALL
```

14.4.10 Cycle 814 TURN PLUNGE LONGITUDINAL EXT.

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run longitudinal turning of shoulders with plunging elements (undercuts). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1. The infeed movement is performed at rapid traverse.
2. The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3. The control retracts the tool at the defined feed rate to the set-up clearance.
4. The control returns the tool at rapid traverse to the cycle starting point.

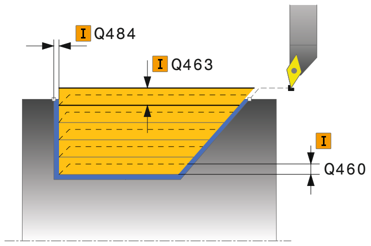
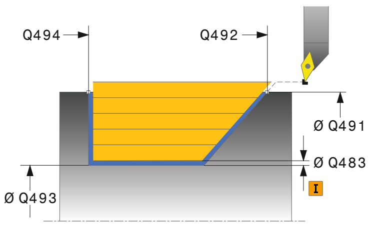
Notes

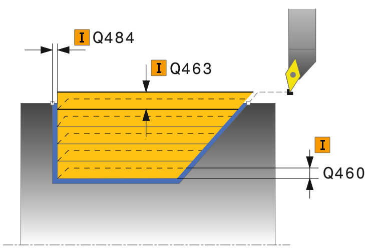
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	**Q492 Contour start in Z?** Z coordinate of the starting point for the plunging path Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle of plunging flank. The reference angle is the line perpendicular to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of face? Angle between the plane surface and the rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
 	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

Example

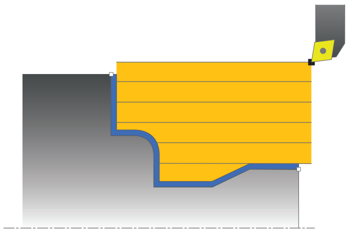
```
11 CYCL DEF 814 TURN PLUNGE LONGITUDINAL EXT. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=-10                ;CONTOUR START IN Z ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-55                ;CONTOUR END IN Z ~
    Q495=+70                ;ANGLE OF SIDE ~
    Q501=+1                 ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5               ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5               ;RADIUS OF CONTOUR EDGE ~
    Q496=+0                 ;ANGLE OF FACE ~
    Q503=+1                 ;TYPE OF END ELEMENT ~
    Q504=+0.5               ;SIZE OF END ELEMENT ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q506=+0                 ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.11 Cycle 810 TURN CONTOUR LONG.

Application

Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run longitudinal turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in longitudinal direction. The longitudinal cut is run paraxially at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

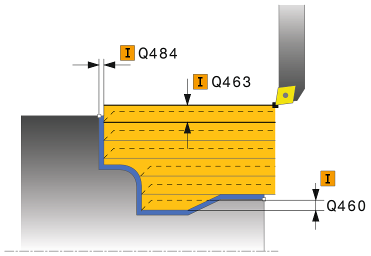
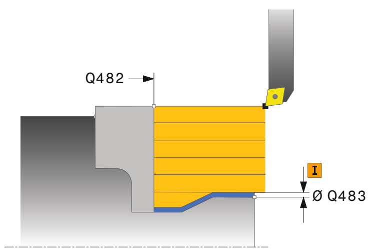
- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

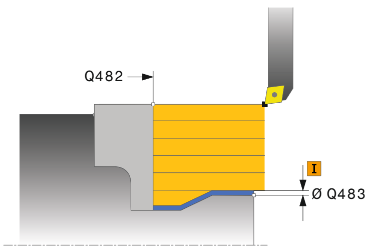
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

Notes on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.
- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q487 Allow plunging (0/1)? Permit the machining of plunging elements: **0**: Do not machine any plunging elements **1**: Machine plunging elements Input: **0**, **1**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	**Q482 Value of cutting limit in Z?** Z value for contour limit Input: **-99999.999...+99999.999**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

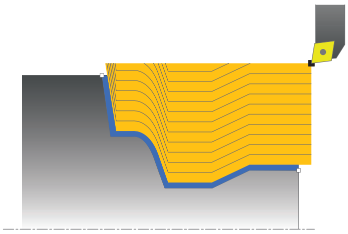
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 810 TURN CONTOUR LONG. ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q499=+0              ;REVERSE CONTOUR ~
    Q463=+3              ;MAX. CUTTING DEPTH ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q487=+1              ;PLUNGE ~
    Q488=+0              ;PLUNGING FEED RATE ~
    Q479=+0              ;CONTOUR MACHINING LIMIT ~
    Q480=+0              ;DIAMETER LIMIT VALUE ~
    Q482=+0              ;LIMIT VALUE Z ~
    Q506=+0              ;CONTOUR SMOOTHING
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  Z-10
20 RND R5
21 L  X+40  Z-35
22 RND R5
23 L  X+50  Z-40
24 L  Z-55
25 CC  X+60  Z-55
26 C  X+60  Z-60
27 L  X+100
28 LBL 0
```

14.4.12 Cycle 815 CONTOUR-PAR. TURNING

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning with roughing is contour-parallel.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and end point. The cut is performed in contour-parallel mode at the defined feed rate **Q478**.
3 The control returns the tool at the defined feed rate back to the starting position in the X coordinate.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

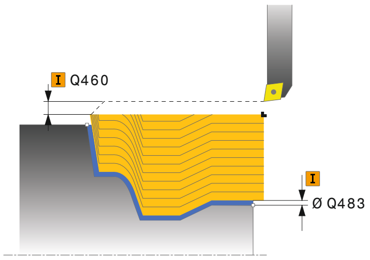
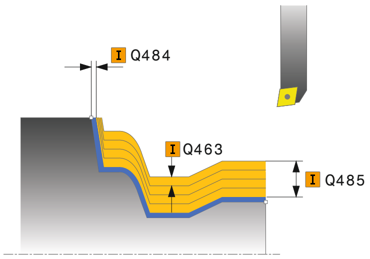
Notes

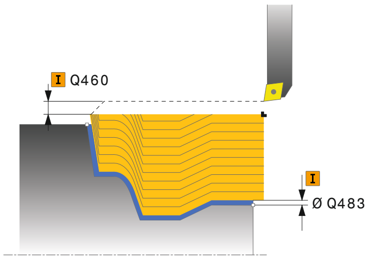
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

Notes on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.
- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
 	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q485 Allowance for workpiece blank? Contour-parallel oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q486 Type of cut lines (=0/1)? Define the type of cutting lines: **0**: Cuts with consistent chip cross section **1**: Equidistance cut distribution Input: **0**, **1**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

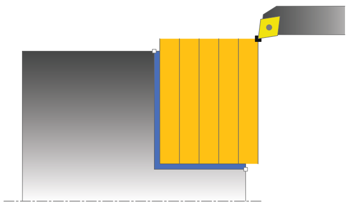
Example

```
11 CYCL DEF 815 CONTOUR-PAR. TURNING ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q485=+5                ;ALLOWANCE ON BLANK ~
    Q486=+0                ;INTERSECTING LINES ~
    Q499=+0                ;REVERSE CONTOUR ~
    Q463=+3                ;MAX. CUTTING DEPTH ~
    Q478=0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.13 Cycle 821 SHOULDER, FACE

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to face turn right-angled shoulders.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run

The cycle machines the area from the cycle starting point to the end point defined in the cycle.

1. The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. The control retracts the tool at the defined feed rate by the infeed value.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control repeats this procedure (steps 1 to 4) until the contour is completed.
6. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1. The control moves the tool in the Z coordinate to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2. The control performs a paraxial infeed movement at rapid traverse.
3. The control finishes the contour of the finished part at the defined feed rate **Q505**.
4. The control retracts the tool at the defined feed rate to the set-up clearance.
5. The control returns the tool at rapid traverse to the cycle starting point.

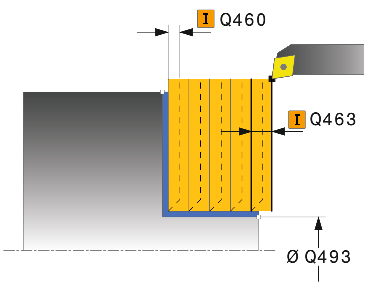
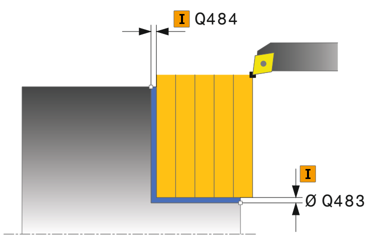
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q463 Maximum cutting depth? Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

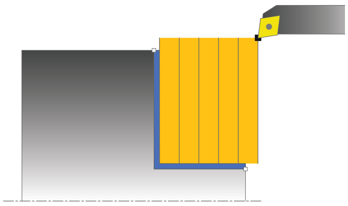
Example

```
11 CYCL DEF 821 SHOULDER, FACE ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q493=+30                ;DIAMETER AT CONTOUR END ~
    Q494=-5                 ;CONTOUR END IN Z ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q506=+0                 ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.14 Cycle 822 SHOULDER, FACE. EXT.

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to face turn shoulders. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the face and circumferential surfaces
- You can insert a radius in the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the starting point is within the area to be machined, the control positions the tool in the Z coordinate and then in the X coordinate to set-up clearance and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control performs a paraxial infeed movement at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

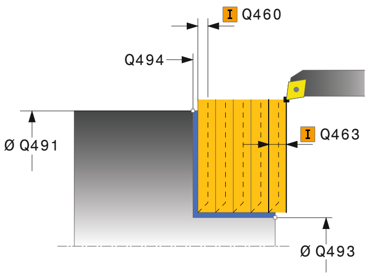
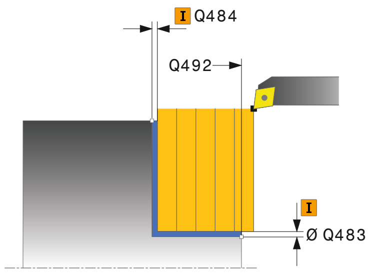
Notes

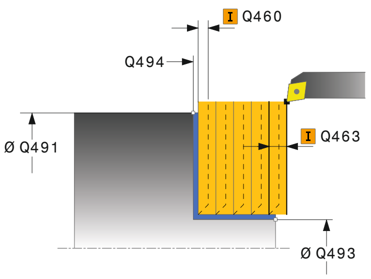
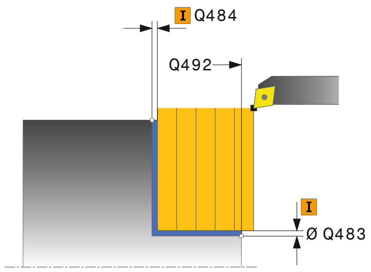
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of the face? Angle between the plane surface and the rotary axis Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of circumferen. surface? Angle between the circumferential surface and rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

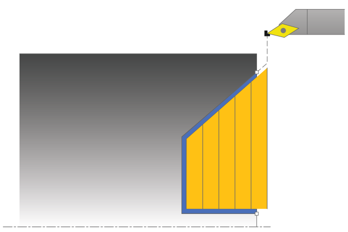
Example

```
11 CYCL DEF 822 SHOULDER, FACE. EXT. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=+0                 ;CONTOUR START IN Z ~
    Q493=+30                ;DIAMETER AT CONTOUR END ~
    Q494=-15                ;CONTOUR END IN Z ~
    Q495=+0                 ;ANGLE OF FACE ~
    Q501=+1                 ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5               ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5               ;RADIUS OF CONTOUR EDGE ~
    Q496=+5                 ;ANGLE OF CIRCUM. SURFACE ~
    Q503=+1                 ;TYPE OF END ELEMENT ~
    Q504=+0.5               ;SIZE OF END ELEMENT ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q506=+0                 ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.15 Cycle 823 TURN TRANSVERSE PLUNGE

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run face turning of plunging elements (undercuts).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in traverse direction at the defined feed rate.
3 The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

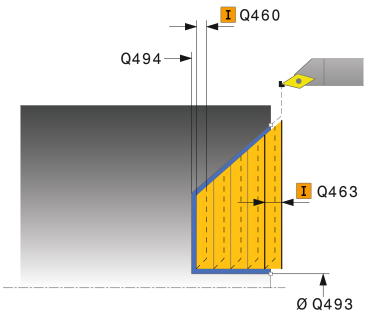
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

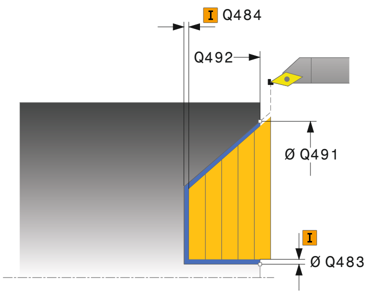
Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic | **Parameter**





Q215 Machining operation (0/1/2/3)?
Define extent of machining:
0: Roughing and finishing
1: Only roughing
2: Only finishing to final dimension
3: Only finishing to oversize
Input: **0**, **1**, **2**, **3**

Q460 Set-up clearance?
Distance for retraction and prepositioning. This value has an incremental effect.
Input: **0...999.999**

Q491 Diameter at contour start?
X coordinate of the contour starting point (diameter value)
Input: **-99999.999...+99999.999**

Q492 Contour start in Z?
Z coordinate of the starting point for the plunging path
Input: **-99999.999...+99999.999**

Q493 Diameter at end of contour?
X coordinate of the contour end point (diameter value)
Input: **-99999.999...+99999.999**

Q494 Contour end in Z?
Z coordinate of the contour end point
Input: **-99999.999...+99999.999**

Q495 Angle of side?
Angle of plunging flank. The reference angle is a line parallel to the rotary axis.
Input: **0...89.9999**

Q463 Maximum cutting depth?
Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts.
Input: **0...99.999**

Q478 Roughing feed rate?
Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute.
Input: **0...99999.999** or **FAUTO**

Q483 Oversize for diameter?
Diameter oversize on the defined contour. This value has an incremental effect.
Input: **0...99.999**

Help graphic	Parameter
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

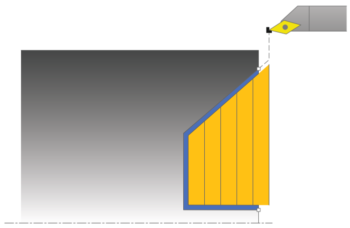
Example

```
11 CYCL DEF 823 TURN TRANSVERSE PLUNGE ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=+0                 ;CONTOUR START IN Z ~
    Q493=+20                ;DIAMETER AT CONTOUR END ~
    Q494=-5                 ;CONTOUR END IN Z ~
    Q495=+60                ;ANGLE OF SIDE ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q506=+0                 ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.16 Cycle 824 TURN PLUNGE TRANSVERSE EXT.

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run face turning of plunging elements (undercuts).
Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define an angle for the face and a radius for the contour edge

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

In undercutting, the control uses feed rate **Q478** for the infeed. The control always retracts the tool to the set-up clearance.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in traverse direction at the defined feed rate.
3 The control retracts the tool at the defined feed rate by the infeed value **Q478**.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

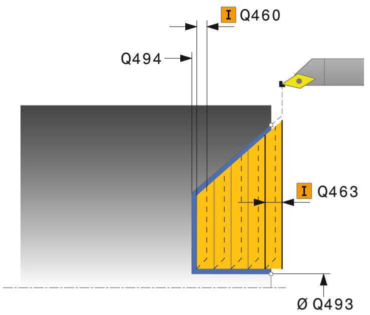
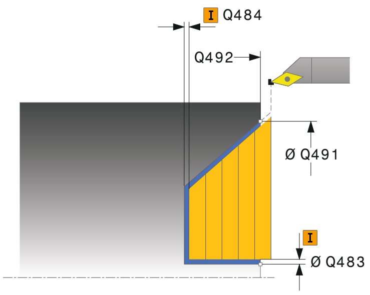
Notes

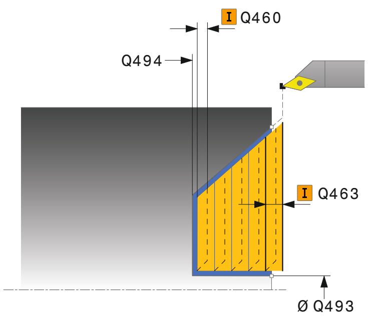
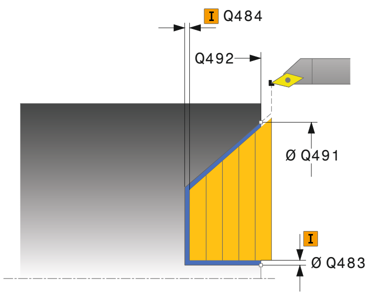
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

Note on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	**Q460 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q491 Diameter at contour start? X coordinate of the starting point for the plunging path (diameter value) Input: **-99999.999...+99999.999**
	**Q492 Contour start in Z?** Z coordinate of the starting point for the plunging path Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle of plunging flank. The reference angle is a line parallel to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of circumferen. surface? Angle between the circumferential surface and rotary axis Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end (plane surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	**Q463 Maximum cutting depth?** Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

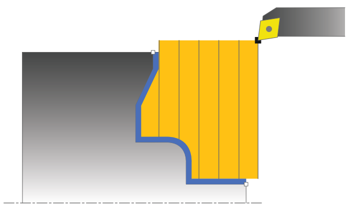
Example

```
11 CYCL DEF 824 TURN PLUNGE TRANSVERSE EXT. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=+0                 ;CONTOUR START IN Z ~
    Q493=+20                ;DIAMETER AT CONTOUR END ~
    Q494=-10                ;CONTOUR END IN Z ~
    Q495=+70                ;ANGLE OF SIDE ~
    Q501=+1                 ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5               ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5               ;RADIUS OF CONTOUR EDGE ~
    Q496=+0                 ;ANGLE OF FACE ~
    Q503=+1                 ;TYPE OF END ELEMENT ~
    Q504=+0.5               ;SIZE OF END ELEMENT ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q506=+0                 ;CONTOUR SMOOTHING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.17 Cycle 820 TURN CONTOUR TRANSV.

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run face turning of workpieces with any turning contours. The contour description is in a subprogram.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 The control performs a paraxial infeed movement at rapid traverse. The control calculates the infeed value based on **Q463 Maximum cutting depth**.
2 The control machines the area between the starting position and the end point in transverse direction. The transverse cut is run paraxially at the defined feed rate **Q478**.
3 The control retracts the tool at the defined feed rate by the infeed value.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control repeats this procedure (steps 1 to 4) until the contour is completed.
6 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1 The infeed movement is performed at rapid traverse.
2 The control finishes the contour of the finished part (contour starting point to contour end point) at the defined feed rate **Q505**.
3 The control retracts the tool at the defined feed rate to the set-up clearance.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

▶ Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- The control takes the cutting geometry of the tool into account to prevent damage to contour elements. If it is not possible to machine the entire workpiece with the active tool, the control will display a warning.
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.
- Also refer to the fundamentals of the turning cycles.
  Further information: "Fundamentals of turning cycles", Page 693

Notes on programming

- Program a positioning block to a safe position with radius compensation **R0** before the cycle call.
- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter

Q215 Machining operation (0/1/2/3)?

Define extent of machining:

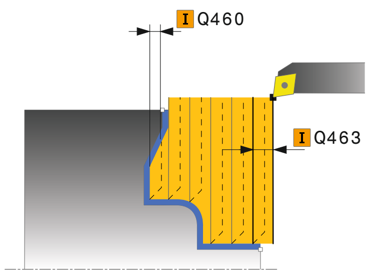
0: Roughing and finishing

1: Only roughing

2: Only finishing to final dimension

3: Only finishing to oversize

Input: **0**, **1**, **2**, **3**



Q460 Set-up clearance?

Distance for retraction and prepositioning. This value has an incremental effect.

Input: **0...999.999**

Q499 Reverse the contour (0-2)?

Define the machining direction of the contour:

0: Contour is executed in the programmed direction

1: Contour is executed in the direction opposite to the programmed direction

2: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted

Input: **0**, **1**, **2**

Q463 Maximum cutting depth?

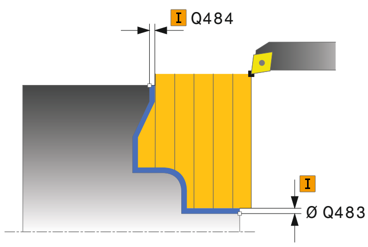
Maximum infeed in the axial direction. The infeed is distributed evenly to avoid abrasive cuts.

Input: **0...99.999**

Q478 Roughing feed rate?

Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute.

Input: **0...99999.999** or **FAUTO**



Q483 Oversize for diameter?

Diameter oversize on the defined contour. This value has an incremental effect.

Input: **0...99.999**

Q484 Oversize in Z?

Oversize of the defined contour in the axial direction. This value has an incremental effect.

Input: **0...99.999**

Q505 Finishing feed rate?

Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute.

Input: **0...99999.999** or **FAUTO**

Help graphic	Parameter
	Q487 Allow plunging (0/1)? Permit the machining of plunging elements: **0**: Do not machine any plunging elements **1**: Machine plunging elements Input: **0**, **1**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**
	Q506 Contour smoothing (0/1/2)? **0**: Along the contour after every cut (within the infeed area) **1**: Contour smoothing after the last cut (entire contour); retract by 45° **2**: No contour smoothing; retract by 45° Input: **0**, **1**, **2**

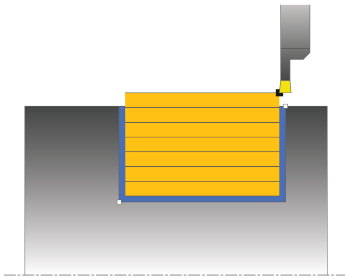
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 820 TURN CONTOUR TRANSV. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q499=+0                 ;REVERSE CONTOUR ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q487=+1                 ;PLUNGE ~
    Q488=+0                 ;PLUNGING FEED RATE ~
    Q479=+0                 ;CONTOUR MACHINING LIMIT ~
    Q480=+0                 ;DIAMETER LIMIT VALUE ~
    Q482=+0                 ;LIMIT VALUE Z ~
    Q506=+0                 ;CONTOUR SMOOTHING
14 L  X+75  Y+0  Z+2 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+75  Z-20
19 L  X+50
20 RND R2
21 L  X+20  Z-25
22 RND R2
23 L  Z+0
24 LBL 0
```

14.4.18 Cycle 841 SIMPLE REC. TURNG., RADIAL DIR.

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

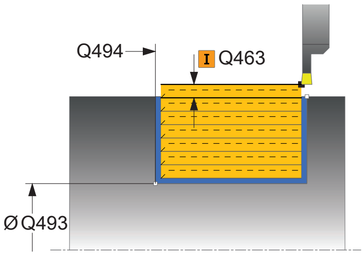
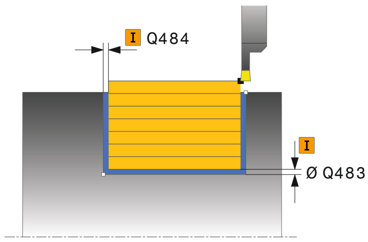
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**

Help graphic	Parameter
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

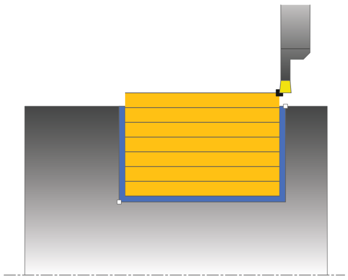
Example

```
11 CYCL DEF 841 SIMPLE REC. TURNG., RADIAL DIR.. ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q493=+50                 ;DIAMETER AT CONTOUR END ~
    Q494=-50                 ;CONTOUR END IN Z ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q463=+2                  ;MAX. CUTTING DEPTH ~
    Q507=+0                  ;MACHINING DIRECTION ~
    Q508=+0                  ;OFFSET WIDTH ~
    Q509=+0                  ;DEPTH COMPENSATION ~
    Q488=+0                  ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.19 Cycle 842 ENH.REC.TURNNG, RAD.

Application

	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to recess right-angled slots in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the X coordinate of the starting point is less than **Q491 Diameter at contour start**, the control positions the tool in the X coordinate to **Q491** and begins the cycle there.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at the cycle call as the cycle starting point. If the X coordinate of the starting point is less than **Q491 DIAMETER AT CONTOUR START**, the control positions the tool in the X coordinate to **Q491** and begins the cycle there.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the control finishes the entire slot in one pass.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

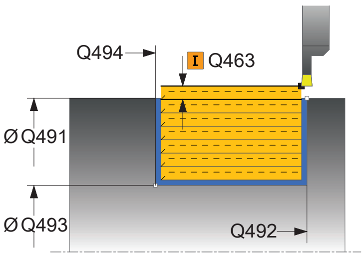
Notes

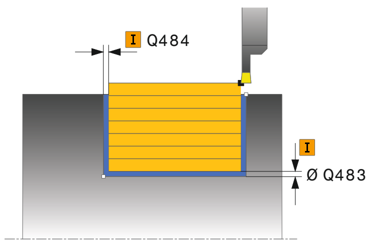
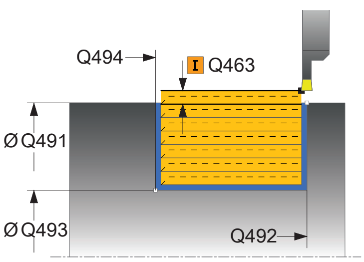
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call (cycle start point) influences the area to be machined.
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q491 Diameter at contour start?** X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and the normal line to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

<table>
<tr><th>Help graphic</th><th>Parameter</th></tr>
<tr><td></td><td>Q496 Angle of second side?
Angle between the edge at the contour end point and the normal line to the rotary axis.
Input: 0...89.9999</td></tr>
<tr><td></td><td>Q503 End element type (0/1/2)?
Define the type of element at the contour end:
0: No additional element
1: Element is a chamfer
2: Element is a radius
Input: 0, 1, 2</td></tr>
<tr><td></td><td>Q504 Size of end element?
Size of the end element (chamfer section)
Input: 0...999.999</td></tr>
<tr><td></td><td>Q478 Roughing feed rate?
Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute.
Input: 0...99999.999 or FAUTO</td></tr>
<tr><td>
</td><td>Q483 Oversize for diameter?
Diameter oversize on the defined contour. This value has an incremental effect.
Input: 0...99.999</td></tr>
<tr><td></td><td>Q484 Oversize in Z?
Oversize of the defined contour in the axial direction. This value has an incremental effect.
Input: 0...99.999</td></tr>
<tr><td></td><td>Q505 Finishing feed rate?
Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute.
Input: 0...99999.999 or FAUTO</td></tr>
<tr><td>
</td><td>Q463 Maximum cutting depth?
Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts.
Input: 0...99.999</td></tr>
<tr><td></td><td>Q507 Direction (0=bidir./1=unidir.)?
Cutting direction:
0: Bidirectional (in both directions)
1: Unidirectional (in direction of contour)
Input: 0, 1</td></tr>
</table>

Help graphic	Parameter
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

Example

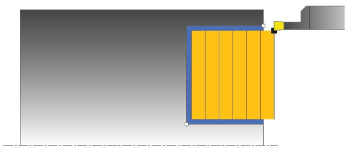
```
11 CYCL DEF 842 EXPND. RECESS, RADL. ~
    Q215=+0          ;MACHINING OPERATION ~
    Q460=+2          ;SAFETY CLEARANCE ~
    Q491=+75         ;DIAMETER AT CONTOUR START ~
    Q492=-20         ;CONTOUR START IN Z ~
    Q493=+50         ;DIAMETER AT CONTOUR END ~
    Q494=-50         ;CONTOUR END IN Z ~
    Q495=+5          ;ANGLE OF SIDE ~
    Q501=+1          ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5        ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5        ;RADIUS OF CONTOUR EDGE ~
    Q496=+5          ;ANGLE OF SECOND SIDE ~
    Q503=+1          ;TYPE OF END ELEMENT ~
    Q504=+0.5        ;SIZE OF END ELEMENT ~
    Q478=+0.3        ;ROUGHING FEED RATE ~
    Q483=+0.4        ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2        ;OVERSIZE IN Z ~
    Q505=+0.2        ;FINISHING FEED RATE ~
    Q463=+2          ;MAX. CUTTING DEPTH ~
    Q507=+0          ;MACHINING DIRECTION ~
    Q508=+0          ;OFFSET WIDTH ~
    Q509=+0          ;DEPTH COMPENSATION ~
    Q488=+0          ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.20 Cycle 851 SIMPLE REC TURNG, AX

Application

Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse is alternatively machined. The machining process thus requires a minimum of retraction and infeed movements.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines the area from the cycle starting point to the end point defined in the cycle.

1 From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2 The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4 If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5 The tool recesses to the next plunging depth.
6 The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control returns the tool at rapid traverse to the cycle starting point.

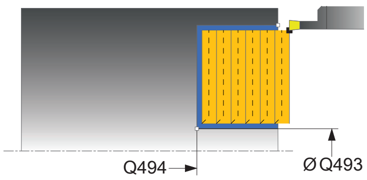
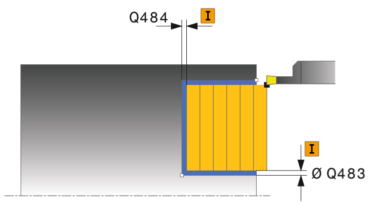
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**

Help graphic	Parameter
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

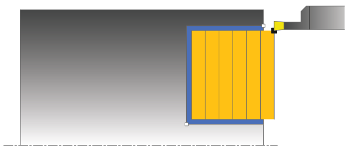
Example

```
11 CYCL DEF 851 SIMPLE REC TURNG, AX ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-10                ;CONTOUR END IN Z ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q463=+2                 ;MAX. CUTTING DEPTH ~
    Q507=+0                 ;MACHINING DIRECTION ~
    Q508=+0                 ;OFFSET WIDTH ~
    Q509=+0                 ;DEPTH COMPENSATION ~
    Q488=+0                 ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.21 Cycle 852 ENH.REC.TURNING, AX.

Application

	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to recess right-angled slots in traverse direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed. The machining process thus requires a minimum of retraction and infeed movements. Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. From the cycle starting point, the control performs a recessing traverse until the first plunging depth is reached.
2. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
3. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
4. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
5. The tool recesses to the next plunging depth.
6. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
7. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
8. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes the slot floor at the defined feed rate. If a radius for contour edges **Q500** was specified, the control finishes the entire slot in one pass.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control returns the tool at rapid traverse to the cycle starting point.

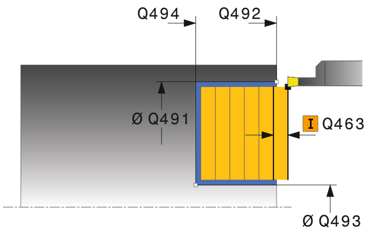
Notes

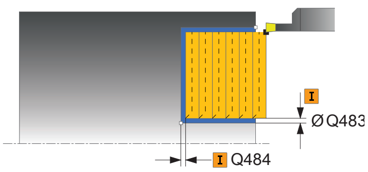
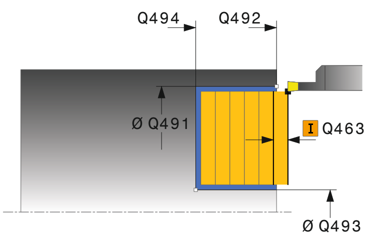
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Note on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q491 Diameter at contour start?** X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and a line parallel to the turning axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge of the contour end point and a line parallel to the turning axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**

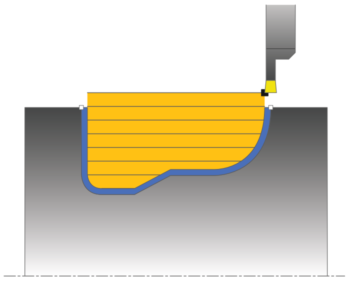
Help graphic	Parameter
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**

Example

```
11 CYCL DEF 852 ENH.REC.TURNING, AX. ~
    Q215=+0              ;MACHINING OPERATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q491=+75             ;DIAMETER AT CONTOUR START ~
    Q492=-20             ;CONTOUR START IN Z ~
    Q493=+50             ;DIAMETER AT CONTOUR END ~
    Q494=-50             ;CONTOUR END IN Z ~
    Q495=+5              ;ANGLE OF SIDE ~
    Q501=+1              ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5            ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5            ;RADIUS OF CONTOUR EDGE ~
    Q496=+5              ;ANGLE OF SECOND SIDE ~
    Q503=+1              ;TYPE OF END ELEMENT ~
    Q504=+0.5            ;SIZE OF END ELEMENT ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2            ;OVERSIZE IN Z ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q463=+2              ;MAX. CUTTING DEPTH ~
    Q507=+0              ;MACHINING DIRECTION ~
    Q508=+0              ;OFFSET WIDTH ~
    Q509=+0              ;DEPTH COMPENSATION ~
    Q488=+0              ;PLUNGING FEED RATE
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.22 Cycle 840 RECESS TURNG, RADIAL

Application



This cycle enables you to recess slots of any form in longitudinal direction. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the X coordinate of the starting point is less than the contour starting point, the control positions the tool in the X coordinate to the contour starting point and begins the cycle there.

1 The control positions the tool at rapid traverse in the Z coordinate (first recessing position).
2 The control performs a recessing traverse until the first plunging depth is reached.
3 The control machines the area between the starting position and the end point in longitudinal direction at the defined feed rate **Q478**.
4 If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5 If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
6 The tool recesses to the next plunging depth.
7 The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8 The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9 The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side walls of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

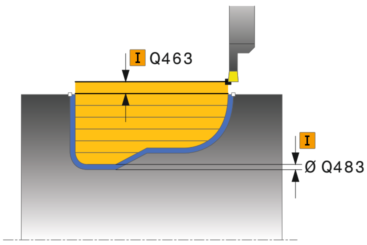
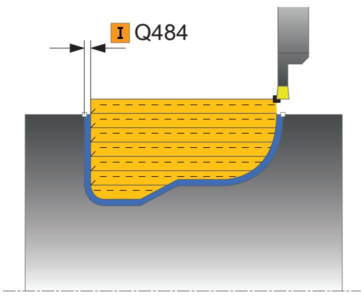
- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	**Q484 Oversize in Z?** Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**

Help graphic	Parameter
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q499 Reverse contour (0=no/1=yes)? Machining direction: **0**: Machining in the direction of contour **1**: Machining in the direction opposite to the contour direction Input: **0**, **1**

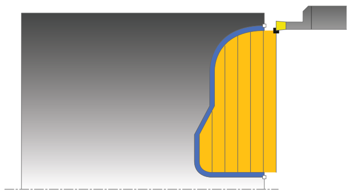
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 840 RECESS TURNG, RADIAL ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q488=+0                ;PLUNGING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE ~
    Q479=+0                ;CONTOUR MACHINING LIMIT ~
    Q480=+0                ;DIAMETER LIMIT VALUE ~
    Q482=+0                ;LIMIT VALUE Z ~
    Q463=+2                ;MAX. CUTTING DEPTH ~
    Q507=+0                ;MACHINING DIRECTION ~
    Q508=+0                ;OFFSET WIDTH ~
    Q509=+0                ;DEPTH COMPENSATION ~
    Q499=+0                ;REVERSE CONTOUR
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z-10
19 L  X+40  Z-15
20 RND R3
21 CR  X+40  Z-35 R+30 DR+
22 RND R3
23 L  X+60  Z-40
24 LBL 0
```

14.4.23 Cycle 850 RECESS TURNG, AXIAL

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to machine slots of any shape in transverse direction by recess turning. With recess turning, a recessing traverse to plunging depth and then a roughing traverse are alternatively performed.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1. The control positions the tool at rapid traverse in the X coordinate (first recessing position).
2. The control performs a recessing traverse until the first plunging depth is reached.
3. The control machines the area between the starting position and the end point in transverse direction at the defined feed rate **Q478**.
4. If the input parameter **Q488** is defined in the cycle, plunging elements are machined at the programmed feed rate for plunging.
5. If only one machining direction **Q507=1** was specified in the cycle, the control lifts off the tool to the set-up clearance, retracts it at rapid traverse and approaches the contour again with the defined feed rate. With machining direction **Q507=0**, infeed is on both sides.
6. The tool recesses to the next plunging depth.
7. The control repeats this procedure (steps 2 to 4) until the slot depth is reached.
8. The control returns the tool to set-up clearance and performs a recessing traverse on both side walls.
9. The control returns the tool at rapid traverse to the cycle starting point.

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side walls of the slot at the defined feed rate **Q505**.
3 The control finishes the slot floor at the defined feed rate.
4 The control returns the tool at rapid traverse to the cycle starting point.

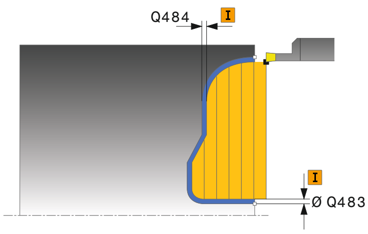
Notes

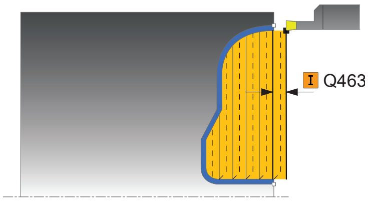
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)
- From the second infeed, the control reduces each further traverse cutting movement by 0.1 mm. This reduces lateral pressure on the tool. If you specified an offset width **Q508** for the cycle, the control reduces the cutting movement by this value. After pre-cutting, the remaining material is removed with a single cut. The control generates an error message if the lateral offset exceeds 80% of the effective cutting width (effective cutting width = cutter width – 2*cutting radius).
- If you programmed a value for **CUTLENGTH**, then it will be taken into account during the roughing operation in this cycle. A message is displayed and the plunging depth is automatically reduced.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q488 Feed rate for plunging (0=auto)? Definition of the feed rate during plunging. This input value is optional. If it is not programmed, then the feed rate defined for turning operations applies. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**

Help graphic	Parameter
	**Q463 Maximum cutting depth?** Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0...99.999**
	Q507 Direction (0=bidir./1=unidir.)? Cutting direction: **0**: Bidirectional (in both directions) **1**: Unidirectional (in direction of contour) Input: **0**, **1**
	Q508 Offset width? Reduction of the cutting length. After pre-cutting, the remaining material is removed with a single cut. If required, the control limits the programmed offset width. Input: **0...99.999**
	Q509 Depth compensat. for finishing? Depending on the material, feed rate, etc., the tool tip is displaced during an operation. You can correct the resulting infeed error with the depth compensation factor. Input: **-9.9999...+9.9999**
	Q499 Reverse contour (0=no/1=yes)? Machining direction: **0**: Machining in the direction of contour **1**: Machining in the direction opposite to the contour direction Input: **0**, **1**

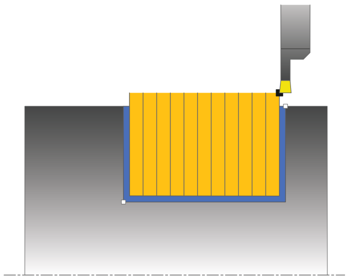
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 850 RECESS TURNG, AXIAL ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q488=0                   ;PLUNGING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q479=+0                  ;CONTOUR MACHINING LIMIT ~
    Q480=+0                  ;DIAMETER LIMIT VALUE ~
    Q482=+0                  ;LIMIT VALUE Z ~
    Q463=+2                  ;MAX. CUTTING DEPTH ~
    Q507=+0                  ;MACHINING DIRECTION ~
    Q508=+0                  ;OFFSET WIDTH ~
    Q509=+0                  ;DEPTH COMPENSATION ~
    Q499=+0                  ;REVERSE CONTOUR
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  Z-10
20 RND R5
21 L  X+40  Y-15
22 L  Z+0
23 LBL 0
```

14.4.24 Cycle 861 SIMPLE RECESS, RADL.

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to radially cut in right-angled slots.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the tool is outside the contour to be machined when the cycle is called, the cycle runs outside machining. If the tool is inside the contour to be machined, the cycle runs inside machining.

Roughing cycle run

The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1 For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2 The control retracts the tool at rapid traverse after each cut
3 The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4 The control machines the remaining material at the feed rate **Q478**
5 The control retracts the tool at rapid traverse after each cut
6 The control repeats steps 4 and 5 until the ridges have been roughed
7 The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle run

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes half the slot width at the defined feed rate.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control finishes half the slot width at the defined feed rate.
8. The control returns the tool at rapid traverse to the cycle starting point.

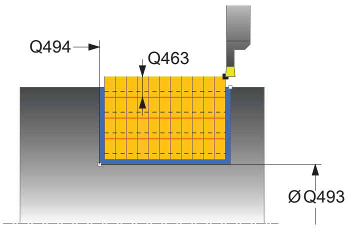
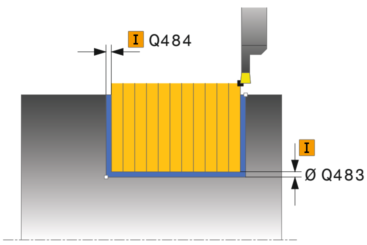
Notes

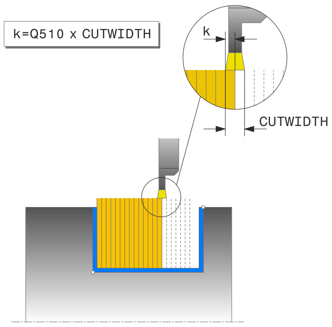
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Limit to plunging depth? Maximum recessing depth per step Input: **0...99.999**

Help graphic	Parameter
	**Q510 Overlap factor for recess width?** Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

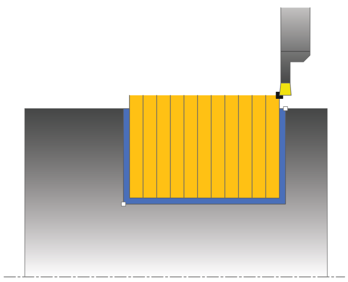
Example

11 CYCL DEF 861 SIMPLE RECESS, RADL. ~	
Q215=+0	;MACHINING OPERATION ~
Q460=+2	;SAFETY CLEARANCE ~
Q493=+50	;DIAMETER AT CONTOUR END ~
Q494=-50	;CONTOUR END IN Z ~
Q478=+0.3	;ROUGHING FEED RATE ~
Q483=+0.4	;OVERSIZE FOR DIAMETER ~
Q484=+0.2	;OVERSIZE IN Z ~
Q505=+0.2	;FINISHING FEED RATE ~
Q463=+0	;LIMIT TO DEPTH ~
Q510=+0.8	;RECESSING OVERLAP ~
Q511=+100	;FEED RATE FACTOR ~
Q462=0	;RETRACTION MODE ~
Q211=3	;DWELL TIME IN REVS ~
Q562=+0	;MULTIPLE PLUNGING
12 L X+75 Y+0 Z+2 FMAX M303	
13 CYCL CALL	

14.4.25 Cycle 862 EXPND. RECESS, RADL.

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to radially cut in slots. Expanded scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the start diameter **Q491** is larger than the end diameter **Q493**, the cycle runs outside machining. If the start diameter **Q491** is less than the end diameter **Q493**, the cycle runs inside machining.

Roughing cycle run

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1. For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2. The control retracts the tool at rapid traverse after each cut
3. The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4. The control machines the remaining material at the feed rate **Q478**
5. The control retracts the tool at rapid traverse after each cut
6. The control repeats steps 4 and 5 until the ridges have been roughed
7. The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle run

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes half the slot width at the defined feed rate.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control finishes half the slot width at the defined feed rate.
8. The control returns the tool at rapid traverse to the cycle starting point.

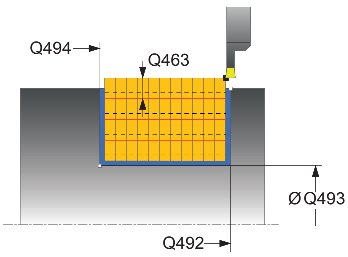
Notes

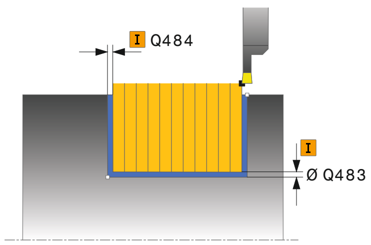
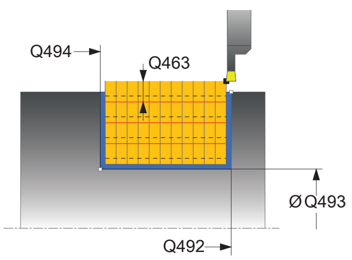
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

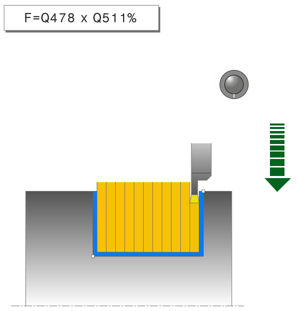
Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	**Q215 Machining operation (0/1/2/3)?** Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and the normal line to the rotary axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge at the contour end point and the normal line to the rotary axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Limit to plunging depth?** Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**

Help graphic	Parameter
	**Q511 Feed rate factor in %?** Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

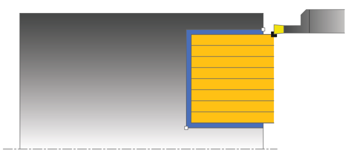
Example

```
11 CYCL DEF 862 EXPND. RECESS, RADL. ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=-20                ;CONTOUR START IN Z ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-50                ;CONTOUR END IN Z ~
    Q495=+5                 ;ANGLE OF SIDE ~
    Q501=+1                 ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5               ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5               ;RADIUS OF CONTOUR EDGE ~
    Q496=+5                 ;ANGLE OF SECOND SIDE ~
    Q503=+1                 ;TYPE OF END ELEMENT ~
    Q504=+0.5               ;SIZE OF END ELEMENT ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q463=+0                 ;LIMIT TO DEPTH ~
    Q510=0.8                ;RECESSING OVERLAP ~
    Q511=+100               ;FEED RATE FACTOR ~
    Q462=+0                 ;RETRACTION MODE ~
    Q211=3                  ;DWELL TIME IN REVS ~
    Q562=+0                 ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.26 Cycle 871 SIMPLE RECESS, AXIAL

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to perform axial recessing of right-angled slots (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. The cycle machines only the area from the cycle starting point to the end point defined in the cycle.

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1. For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2. The control retracts the tool at rapid traverse after each cut
3. The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4. The control machines the remaining material at the feed rate **Q478**
5. The control retracts the tool at rapid traverse after each cut
6. The control repeats steps 4 and 5 until the ridges have been roughed
7. The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle run

1. The control positions the tool at rapid traverse to the first slot side.
2. The control finishes the side wall of the slot at the defined feed rate **Q505**.
3. The control finishes half the slot width at the defined feed rate.
4. The control retracts the tool at rapid traverse.
5. The control positions the tool at rapid traverse to the second slot side.
6. The control finishes the side wall of the slot at the defined feed rate **Q505**.
7. The control finishes half the slot width at the defined feed rate.
8. The control returns the tool at rapid traverse to the cycle starting point.

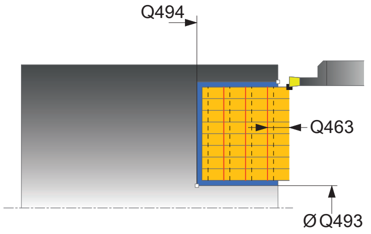
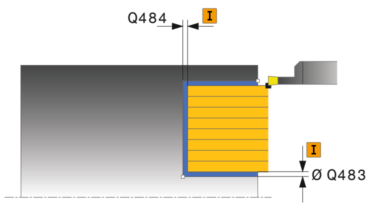
Notes

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
 	**Q493 Diameter at end of contour?** X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
 	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q463 Limit to plunging depth? Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**

Help graphic	Parameter
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

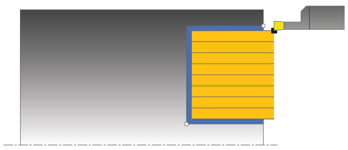
Example

```
11 CYCL DEF 871 SIMPLE RECESS, AXIAL ~
    Q215=+0                ;MACHINING OPERATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q493=+50               ;DIAMETER AT CONTOUR END ~
    Q494=-10               ;CONTOUR END IN Z ~
    Q478=+0.3              ;ROUGHING FEED RATE ~
    Q483=+0.4              ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2              ;OVERSIZE IN Z ~
    Q505=+0.2              ;FINISHING FEED RATE ~
    Q463=+0                ;LIMIT TO DEPTH ~
    Q510=+0,8              ;RECESSING OVERLAP ~
    Q511=+100              ;FEED RATE FACTOR ~
    Q462=0                 ;RETRACTION MODE ~
    Q211=3                 ;DWELL TIME IN REVS ~
    Q562=+0                ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.27 Cycle 872 EXPND. RECESS, AXIAL

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to perform axial recessing of slots (face recessing). Extended scope of function:

- You can insert a chamfer or curve at the contour start and contour end.
- In the cycle you can define angles for the side walls of the slot
- You can insert radii in the contour edges

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1 For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2 The control retracts the tool at rapid traverse after each cut
3 The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4 The control machines the remaining material at the feed rate **Q478**
5 The control retracts the tool at rapid traverse after each cut
6 The control repeats steps 4 and 5 until the ridges have been roughed
7 The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point. If the Z coordinate of the starting point is less than **Q492 Contour start in Z**, the control positions the tool in the Z coordinate to **Q492** and begins the cycle there.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control retracts the tool at rapid traverse.
4 The control positions the tool at rapid traverse to the second slot side.
5 The control finishes the side wall of the slot at the defined feed rate **Q505**.
6 The control finishes one half of the slot at the defined feed rate.
7 The control positions the tool at rapid traverse to the first side.
8 The control finishes the other half of the slot at the defined feed rate.
9 The control returns the tool at rapid traverse to the cycle starting point.

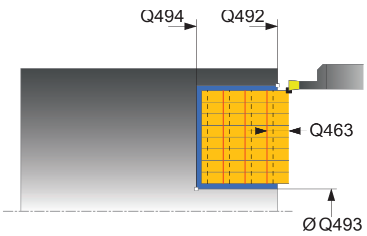
Notes

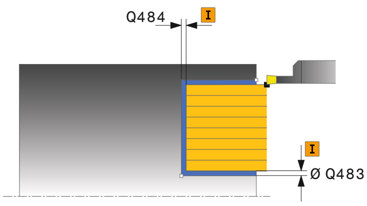
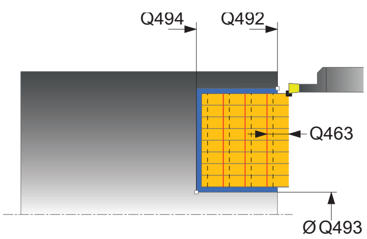
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	**Q492 Contour start in Z?** Z coordinate of the contour starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the contour end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the contour end point Input: **-99999.999...+99999.999**
	Q495 Angle of side? Angle between the edge of the contour starting point and a line parallel to the turning axis. Input: **0...89.9999**
	Q501 Starting element type (0/1/2)? Define the type of element at the beginning of the contour (circumferential surface): **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q502 Size of starting element? Size of the starting element (chamfer section) Input: **0...999.999**
	Q500 Radius of the contour corner? Radius of the inside corner of the contour. If no radius is specified, the radius will be that of the indexable insert. Input: **0...999.999**

Help graphic	Parameter
	Q496 Angle of second side? Angle between the edge of the contour end point and a line parallel to the turning axis. Input: **0...89.9999**
	Q503 End element type (0/1/2)? Define the type of element at the contour end: **0**: No additional element **1**: Element is a chamfer **2**: Element is a radius Input: **0**, **1**, **2**
	Q504 Size of end element? Size of the end element (chamfer section) Input: **0...999.999**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q463 Limit to plunging depth?** Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**

Help graphic	Parameter
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

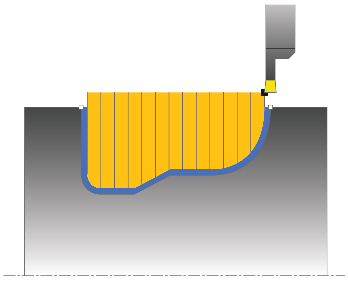
Example

```
11 CYCL DEF 872 EXPND. RECESS, AXIAL ~
    Q215=+0                 ;MACHINING OPERATION ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q491=+75                ;DIAMETER AT CONTOUR START ~
    Q492=-20                ;CONTOUR START IN Z ~
    Q493=+50                ;DIAMETER AT CONTOUR END ~
    Q494=-50                ;CONTOUR END IN Z ~
    Q495=+5                 ;ANGLE OF SIDE ~
    Q501=+1                 ;TYPE OF STARTING ELEMENT ~
    Q502=+0.5               ;SIZE OF STARTING ELEMENT ~
    Q500=+1.5               ;RADIUS OF CONTOUR EDGE ~
    Q496=+5                 ;ANGLE OF SECOND SIDE ~
    Q503=+1                 ;TYPE OF END ELEMENT ~
    Q504=+0.5               ;SIZE OF END ELEMENT ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q483=+0.4               ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2               ;OVERSIZE IN Z ~
    Q505=+0.2               ;FINISHING FEED RATE ~
    Q463=+0                 ;LIMIT TO DEPTH ~
    Q510=+0.08              ;RECESSING OVERLAP ~
    Q511=+100               ;FEED RATE FACTOR ~
    Q462=+0                 ;RETRACTION MODE ~
    Q211=+3                 ;DWELL TIME IN REVS ~
    Q562=+0                 ;MULTIPLE PLUNGING
12 L  X+75  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.28 Cycle 860 CONT. RECESS, RADIAL

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to radially cut in slots of any form.

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

The cycle can be used for inside and outside machining. If the coordinate of the contour starting point is larger than that of the contour end point, the cycle runs outside machining. If the coordinate of the contour starting point is less than that of the contour end point, the cycle runs inside machining.

Roughing cycle run

1. For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2. The control retracts the tool at rapid traverse.
3. The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4. The control then recesses again, this time with the feed rate **Q478**
5. The control retracts the tool as defined in parameter **Q462**
6. The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7. As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1. For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2. The control retracts the tool at rapid traverse after each cut
3. The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4. The control machines the remaining material at the feed rate **Q478**
5. The control retracts the tool at rapid traverse after each cut
6. The control repeats steps 4 and 5 until the ridges have been roughed
7. The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle run

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes one half of the slot at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes the other half of the slot at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

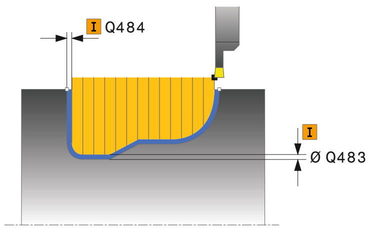
▶ Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

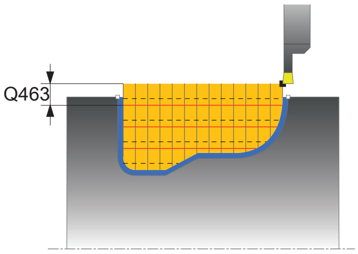
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Feed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q484 Oversize in Z? Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**

Help graphic	Parameter
	**Q463 Limit to plunging depth?** Maximum recessing depth per step Input: **0...99.999**
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

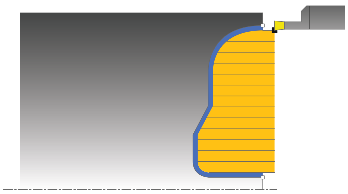
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 860 CONT. RECESS, RADIAL ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q479=+0                  ;CONTOUR MACHINING LIMIT ~
    Q480=+0                  ;DIAMETER LIMIT VALUE ~
    Q482=+0                  ;LIMIT VALUE Z ~
    Q463=+0                  ;LIMIT TO DEPTH ~
    Q510=0.08                ;RECESSING OVERLAP ~
    Q511=+100                ;FEED RATE FACTOR ~
    Q462=+0                  ;RETRACTION MODE ~
    Q211=3                   ;DWELL TIME IN REVS ~
    Q562=+0                  ;MULTIPLE PLUNGING
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z-20
19 L  X+45
20 RND R2
21 L  X+40  Y-25
22 L  Z+0
23 LBL 0
```

14.4.29 Cycle 870 CONT. RECESS, AXIAL

Application

	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to perform axial recessing of slots of any form (face recessing).

You can use the cycle either for roughing, finishing or complete machining. Turning is run paraxially with roughing.

Roughing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to the contour starting point and begins the cycle there.

1 For the first recess with full contact, the control moves the tool at the reduced feed rate **Q511** to the depth of the plunge + allowance.
2 The control retracts the tool at rapid traverse.
3 The control performs a stepover by **Q510** x tool width (**Cutwidth**).
4 The control then recesses again, this time with the feed rate **Q478**
5 The control retracts the tool as defined in parameter **Q462**
6 The control machines the area between the starting position and the end point by repeating steps 2 through 4.
7 As soon as the slot width has been achieved, the control returns the tool at rapid traverse to the cycle starting point.

Multiple plunging

1 For the recess with full contact, the control moves the tool at a reduced feed rate **Q511** to the depth of the plunge + allowance
2 The control retracts the tool at rapid traverse after each cut
3 The position and number of full cuts depend on **Q510** and the width of the tooth (**CUTWIDTH**). Steps 1 to 2 are repeated until all full cuts have been made
4 The control machines the remaining material at the feed rate **Q478**
5 The control retracts the tool at rapid traverse after each cut
6 The control repeats steps 4 and 5 until the ridges have been roughed
7 The control then positions the tool at rapid traverse back to the cycle starting point

Finishing cycle run

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse to the first slot side.
2 The control finishes the side wall of the slot at the defined feed rate **Q505**.
3 The control finishes one half of the slot at the defined feed rate.
4 The control retracts the tool at rapid traverse.
5 The control positions the tool at rapid traverse to the second slot side.
6 The control finishes the side wall of the slot at the defined feed rate **Q505**.
7 The control finishes the other half of the slot at the defined feed rate.
8 The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE

Danger of collision!

The cutting limit defines the contour range to be machined. The approach and departure paths can cross over the cutting limits. The tool position before the cycle call influences the execution of the cutting limit. The TNC7 machines the area to the right or to the left of the cutting limit, depending on which side the tool was positioned before calling the cycle.

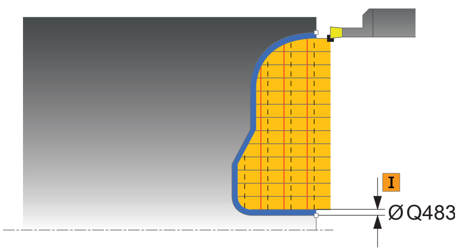
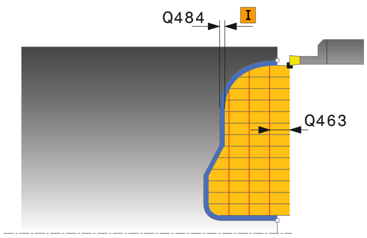
- Before calling the cycle, make sure to position the tool at the side of the cutting boundary (cutting limit) where the material will be machined

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The tool position at cycle call defines the size of the area to be machined (cycle starting point)

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- **FUNCTION TURNDATA CORR TCS: Z/X DCW** and/or an entry in the DCW column of the turning tool table can be used to activate an oversize for the recessing width. DCW can accept positive and negative values and is added to the recessing width: CUTWIDTH + DCWTab + FUNCTION TURNDATA CORR TCS: Z/X DCW. A DCW programmed via **FUNCTION TURNDATA CORR TCS** is not visible while a DCW entered in the table is active in the graphics.
- If multiple plunging is active (**Q562** = **1**) and the value **Q462 RETRACTION MODE** is not equal to 0, then the control issues an error message.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q460 Set-up clearance? Reserved; currently no functionality
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	**Q483 Oversize for diameter?** Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	**Q484 Oversize in Z?** Oversize of the defined contour in the axial direction. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q479 Machining limits (0/1)? Activate cutting limit: **0**: No cutting limit active **1**: Cutting limit (**Q480/Q482**) Input: **0**, **1**
	Q480 Value of diameter limit? X value for contour limit (diameter value) Input: **-99999.999...+99999.999**
	Q482 Value of cutting limit in Z? Z value for contour limit Input: **-99999.999...+99999.999**
	Q463 Limit to plunging depth? Maximum recessing depth per step Input: **0...99.999**

Help graphic	Parameter
	Q510 Overlap factor for recess width? Factor **Q510** influences the lateral infeed of the tool during roughing. **Q510** is multiplied by the **CUTWIDTH** of the tool. This results in the lateral infeed factor "k". Input: **0.001...1**
	Q511 Feed rate factor in %? Factor **Q511** influences the feed rate for full recessing, i.e. when a recess is cut with the entire tool width **CUTWIDTH**. If you use this feed rate factor, optimum cutting conditions can be created during the remaining roughing process. In this manner, you can define the roughing feed rate **Q478** to be so high that it permits optimum cutting conditions for each overlap of the cutting width (**Q510**). The control thus reduces the feed rate by the factor **Q511** only when recessing with full contact. In sum, this can lead to reduced machining times. Input: **0.001...150**
	Q462 Retraction behavior (0/1)? With **Q462**, you define the retraction behavior after the recess. **0**: The control retracts the tool along the contour **1**: The control first moves the tool at an angle away from the contour and then retracts it Input: **0**, **1**
	Q211 Dwell time / 1/min? A dwell time can be specified in revolutions of the tool spindle, which delays the retraction after the recessing on the floor. Retraction is only performed after the tool has remained for **Q211** revolutions. Input: **0...999.99**
	Q562 Multiple plunging (0/1)? **0**: No multiple plunging: the first recess is made into the uncut material, and the subsequent ones are laterally offset and overlap by the amount **Q510** * Width of the cutter (**CUTWIDTH**) **1**: Multiple plunging; rough grooving is performed with full tool engagement into uncut material. Then the remaining ridges are machined. These are recessed successively. This leads to a centralized chip removal, considerably reducing the risk of chip entrapment Input: **0**, **1**

Example

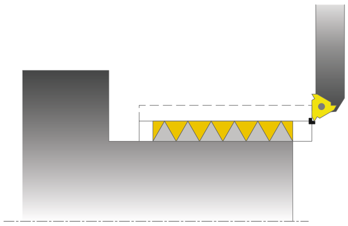
```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 870 CONT. RECESS, AXIAL ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q484=+0.2                ;OVERSIZE IN Z ~
    Q505=+0.2                ;FINISHING FEED RATE ~
    Q479=+0                  ;CONTOUR MACHINING LIMIT ~
    Q480=+0                  ;DIAMETER LIMIT VALUE ~
    Q482=+0                  ;LIMIT VALUE Z ~
    Q463=+0                  ;LIMIT TO DEPTH ~
    Q510=+0.8                ;RECESSING OVERLAP ~
    Q511=+100                ;FEED RATE FACTOR ~
    Q462=+0                  ;RETRACTION MODE ~
    Q211=+3                  ;DWELL TIME IN REVS ~
    Q562=+0                  ;MULTIPLE PLUNGING
14 L  X+75  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  Z-10
20 RND R5
21 L  X+40  Y-15
22 L  Z+0
23 LBL 0
```

14.4.30 Cycle 831 THREAD LONGITUDINAL

Application

Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run longitudinal turning of threads.

You can machine single threads or multi-threads with this cycle.

If you do not enter a thread depth, the cycle uses thread depth in accordance with the ISO1502 standard.

The cycle can be used for inside and outside machining.

Cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control performs a paraxial longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

While the control cuts a thread, the feed-rate override knob is disabled. The feed-rate override knob is still active to a limited extent,

Notes

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g., to machine internal and external threads with only one tool. If, with such a machine, you wish to use an outside tool for inside machining, you can execute machining in the negative X diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position.

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Danger of collision!

If you program an angle of infeed **Q467** wider than the side angle of the thread may destroy the thread flank. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

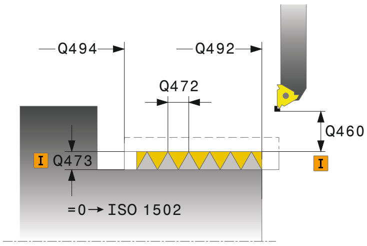
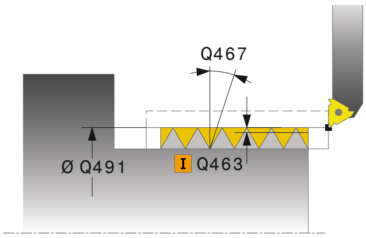
- Do not program the infeed angle **Q467** to be larger than the thread edge angle

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- The number of threads for thread cutting is limited to 500.
- In Cycle **832 THREAD EXTENDED**, parameters are available for approach and overrun.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- The control uses the set-up clearance **Q460** as approach length. The approach path must be long enough for the feed axes to be accelerated to the required velocity.
- The control uses the thread pitch as idle travel path. The idle travel distance must be long enough to decelerate the feed axes.
- If the **TYPE OF INFEED Q468** is equal to 0 (consistent chip cross section), then an **ANGLE OF INFEED** must be defined to be larger than 0 in **Q467**.

Cycle parameters

Help graphic	Parameter
	**Q471 Thread position (0=ext./1=int.)?** Define the position of the thread: **0**: External thread **1**: Internal thread Input: **0**, **1**
	Q460 Setup clearance? Set-up clearance in radial and axial direction. In axial direction, the set-up clearance is used for acceleration (approach path) until the synchronized feed rate is reached. Input: **0...999.999**
	Q491 Thread diameter? Define the nominal diameter of the thread. Input: **0.001...99999.999**,
	Q472 Thread pitch? Pitch of the thread Input: **0...99999.999**
	Q473 Thread depth (radius)? Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch. This value has an incremental effect. Input: **0...999.999**
	Q492 Contour start in Z? Z coordinate of the starting point Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the end point, including the thread runout **Q474** Input: **-99999.999...+99999.999**
	Q474 Length of thread runout? Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**. This value has an incremental effect. Input: **0...999.999**
	Q463 Maximum cutting depth? Maximum plunging depth in radial direction relative to the radius. Input: **0,001...999.999**
	Q467 Feed angle? Angle at which the infeed **Q463** occurs. The reference angle is the line perpendicular to the rotary axis. Input: **0...60**

Help graphic	Parameter
	Q468 Infeed type (0/1)? Define the type of infeed: **0**: Consistent chip cross section (the infeed becomes less as the depth increases) **1**: Constant plunging depth Input: **0**, **1**
	Q470 Starting angle? Angle of the turning spindle at which the thread is to be started. Input: **0...359999**
	Q475 Number of thread grooves? Number of thread grooves Input: **1...500**
	Q476 Number of air cuts? Number of air cuts without infeed at finished thread depth Input: **0...255**

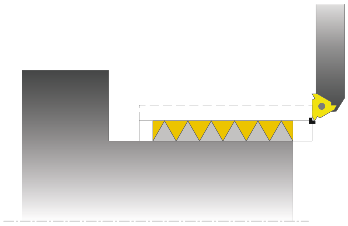
Example

```
11 CYCL DEF 831 THREAD LONGITUDINAL ~
    Q471=+0                 ;THREAD POSITION ~
    Q460=+5                 ;SAFETY CLEARANCE ~
    Q491=+75                ;THREAD DIAMETER ~
    Q472=+2                 ;THREAD PITCH ~
    Q473=+0                 ;DEPTH OF THREAD ~
    Q492=+0                 ;CONTOUR START IN Z ~
    Q494=-15                ;CONTOUR END IN Z ~
    Q474=+0                 ;THREAD RUN-OUT ~
    Q463=+0.5               ;MAX. CUTTING DEPTH ~
    Q467=+30                ;ANGLE OF INFEED ~
    Q468=+0                 ;TYPE OF INFEED ~
    Q470=+0                 ;STARTING ANGLE ~
    Q475=+30                ;NUMBER OF STARTS ~
    Q476=+30                ;NUMBER OF AIR CUTS
12 L  X+80  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.31 Cycle 832 THREAD EXTENDED

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run both face turning and longitudinal turning of threads or tapered threads. Expanded scope of function:

- Selection of a longitudinal thread or transversal thread
- The parameters for dimension type of taper, taper angle, and contour starting point X enable the definition of various tapered threads
- The parameters for the approach length and the idle travel distance define a path in which feed axes can be accelerated and decelerated

You can process single threads or multi-threads with the cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.

Cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1 The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2 The control performs a longitudinal cut. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3 The control retracts the tool at rapid traverse to the set-up clearance.
4 The control returns the tool at rapid traverse to the beginning of cut.
5 The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6 The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7 The control performs the number of air cuts as defined in **Q476**.
8 The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9 The control returns the tool at rapid traverse to the cycle starting point.

While the control cuts a thread, the feed-rate override knob is disabled.
The feed-rate override knob is still active to a limited extent,

Notes

NOTICE

Danger of collision!

If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece.

- With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g., to machine internal and external threads with only one tool. If, with such a machine, you wish to use an outside tool for inside machining, you can execute machining in the negative X diameter range and reverse the direction of workpiece rotation.

NOTICE

Danger of collision!

The retraction motion is directly to the starting position.

- Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

NOTICE

Danger of collision!

If you program an angle of infeed **Q467** wider than the side angle of the thread may destroy the thread flank. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves.

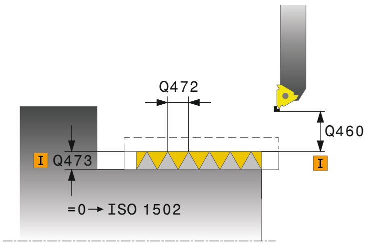
- Do not program the infeed angle **Q467** to be larger than the thread edge angle

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.
- The overrun path (**Q466**) must be long enough to decelerate the feed axes.
- If the **TYPE OF INFEED Q468** is equal to 0 (consistent chip cross section), then an **ANGLE OF INFEED** must be defined to be larger than 0 in **Q467**.

Cycle parameters

Help graphic	Parameter
	Q471 Thread position (0=ext./1=int.)? Define the position of the thread: **0**: External thread **1**: Internal thread Input: **0**, **1**
	Q461 Thread orientation (0/1)? Define the direction of the thread pitch: **0**: L (parallel to the turning axis) **1**: Perpendicular (perpendicular to the turning axis) Input: **0**, **1**
	**Q460 Set-up clearance?** Set-up clearance perpendicular to the thread pitch Input: **0...999.999**
	Q472 Thread pitch? Pitch of the thread Input: **0...99999.999**
	Q473 Thread depth (radius)? Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch. This value has an incremental effect. Input: **0...999.999**
	Q464 Dimens. type taper (0-4)? Type of dimensioning of the taper contour: **0**: Via start and end point **1**: Via end point, start X and angle of taper **2**: Via end point, start Z and angle of taper **3**: Via start point, end X and angle of taper **4**: Via start point, end Z and angle of taper Input: **0**, **1**, **2**, **3**, **4**
	Q491 Diameter at contour start? X coordinate of the contour starting point (diameter value) Input: **-99999.999...+99999.999**
	Q492 Contour start in Z? Z coordinate of the starting point Input: **-99999.999...+99999.999**
	Q493 Diameter at end of contour? X coordinate of the end point (diameter value) Input: **-99999.999...+99999.999**
	Q494 Contour end in Z? Z coordinate of the end point Input: **-99999.999...+99999.999**

Help graphic	Parameter
	Q469 Taper angle (diameter)? Taper angle of the contour Input: **-180...+180**
	Q474 Length of thread runout? Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**. This value has an incremental effect. Input: **0...999.999**
	Q465 Starting path? Length of the path in the direction of the pitch at which the feed axes are accelerated to the required speed. The approach path is outside of the defined thread contour. This value has an incremental effect. Input: **0.1...99.9**
	Q466 Overrun path? Input: **0.1...99.9**
	Q463 Maximum cutting depth? Maximum infeed perpendicular to the thread pitch Input: **0,001...999.999**
	Q467 Feed angle? Angle at which the infeed **Q463** occurs. The reference angle is formed by the parallel line to the thread pitch. Input: **0...60**
	Q468 Infeed type (0/1)? Define the type of infeed: **0**: Consistent chip cross section (the infeed becomes less as the depth increases) **1**: Constant plunging depth Input: **0**, **1**
	Q470 Starting angle? Angle of the turning spindle at which the thread is to be started. Input: **0...359999**
	Q475 Number of thread grooves? Number of thread grooves Input: **1...500**
	Q476 Number of air cuts? Number of air cuts without infeed at finished thread depth Input: **0...255**

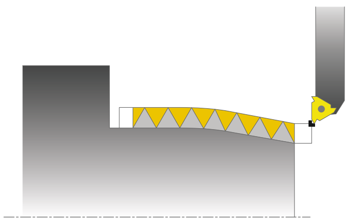
Example

```
11 CYCL DEF 832 THREAD EXTENDED ~
    Q471=+0              ;THREAD POSITION ~
    Q461=+0              ;THREAD ORIENTATION ~
    Q460=+2              ;SAFETY CLEARANCE ~
    Q472=+2              ;THREAD PITCH ~
    Q473=+0              ;DEPTH OF THREAD ~
    Q464=+0              ;DIMENSION TYPE TAPER ~
    Q491=+100            ;DIAMETER AT CONTOUR START ~
    Q492=+0              ;CONTOUR START IN Z ~
    Q493=+110            ;DIAMETER AT CONTOUR END ~
    Q494=-35             ;CONTOUR END IN Z ~
    Q469=+0              ;TAPER ANGLE ~
    Q474=+0              ;THREAD RUN-OUT ~
    Q465=+4              ;STARTING PATH ~
    Q466=+4              ;OVERRUN PATH ~
    Q463=+0.5            ;MAX. CUTTING DEPTH ~
    Q467=+30             ;ANGLE OF INFEED ~
    Q468=+0              ;TYPE OF INFEED ~
    Q470=+0              ;STARTING ANGLE ~
    Q475=+30             ;NUMBER OF STARTS ~
    Q476=+30             ;NUMBER OF AIR CUTS
12 L  X+80  Y+0  Z+2 FMAX M303
13 CYCL CALL
```

14.4.32 Cycle 830 THREAD CONTOUR-PARALLEL

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to run both face turning and longitudinal turning of threads with any shape.

You can machine single threads or multi-threads with this cycle.

If you do not enter a thread depth in the cycle, the cycle uses a standardized thread depth.

The cycle can be used for inside and outside machining.

Cycle sequence

The control uses the position of the tool at cycle call as the cycle starting point.

1. The control positions the tool at rapid traverse at set-up clearance in front of the thread and performs an infeed movement.
2. The control runs a thread cut parallel to the defined thread contour. When doing so, the control synchronizes feed rate and speed so that the defined pitch is machined.
3. The control retracts the tool at rapid traverse to the set-up clearance.
4. The control returns the tool at rapid traverse to the beginning of cut.
5. The control performs an infeed movement. For the infeeds, to the angle of infeed **Q467** is used.
6. The control repeats this procedure (steps 2 to 5) until the thread depth is reached.
7. The control performs the number of air cuts as defined in **Q476**.
8. The control repeats this procedure (steps 2 to 7) until the desired Number of thread grooves **Q475** is reached.
9. The control returns the tool at rapid traverse to the cycle starting point.

> While the control cuts a thread, the feed-rate override knob is disabled. The feed-rate override knob is still active to a limited extent,

Notes

NOTICE
Danger of collision! Cycle **830** runs the overrun **Q466** following the programmed contour. Take the available space into account. ▶ Clamp the workpiece in such a way that there is no danger of collision if the control extends the contour by **Q466**, **Q467**.

NOTICE
Danger of collision! If the tool is pre-positioned at a negative diameter position, the effect of parameter **Q471** Thread position is reversed. This means that the external thread is 1 and the internal thread 0. There is a risk of collision between tool and workpiece. ▶ With some machine types, the turning tool is not clamped in the milling spindle, but in a separate holder adjacent to the spindle. In such cases, the turning tool cannot be rotated through 180°, e.g., to machine internal and external threads with only one tool. If, with such a machine, you wish to use an outside tool for inside machining, you can execute machining in the negative X diameter range and reverse the direction of workpiece rotation.

NOTICE
Danger of collision! The retraction motion is directly to the starting position. ▶ Always position the tool in such a way that the control can approach the starting point at the end of the cycle without collisions.

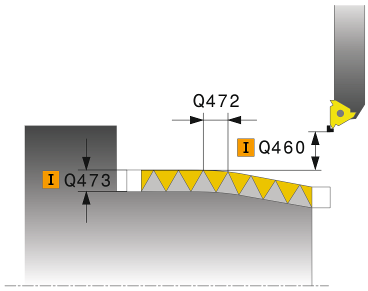
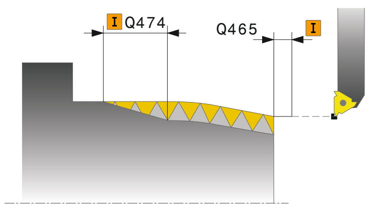
NOTICE
Danger of collision! If you program an angle of infeed **Q467** wider than the side angle of the thread may destroy the thread flank. If the angle of infeed is modified, the position of the thread is shifted in an axial direction. With a changed angle of infeed, the tool can no longer interface the thread grooves. ▶ Do not program the infeed angle **Q467** to be larger than the thread edge angle

- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- Both the approach and overrun take place outside the defined contour.

Notes on programming

- Program a positioning block to the starting position with radius compensation **R0** before the cycle call.
- The approach path (**Q465**) must be long enough for the feed axes to be accelerated to the required velocity.
- The overrun path (**Q466**) must be long enough to decelerate the feed axes.
- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- If the **TYPE OF INFEED Q468** is equal to 0 (consistent chip cross section), then an **ANGLE OF INFEED** must be defined to be larger than 0 in **Q467**.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q471 Thread position (0=ext./1=int.)? Define the position of the thread: **0**: External thread **1**: Internal thread Input: **0**, **1**
	Q461 Thread orientation (0/1)? Define the direction of the thread pitch: **0**: L (parallel to the turning axis) **1**: Perpendicular (perpendicular to the turning axis) Input: **0**, **1**
	**Q460 Set-up clearance?** Set-up clearance perpendicular to the thread pitch Input: **0...999.999**
	Q472 Thread pitch? Pitch of the thread Input: **0...99999.999**
	Q473 Thread depth (radius)? Depth of the thread. If you enter 0, the depth is assumed for a metric thread based on the pitch. This value has an incremental effect. Input: **0...999.999**
	**Q474 Length of thread runout?** Length of the path on which, at the end of the thread, the tool is lifted from the current plunging depth to the thread diameter **Q460**. This value has an incremental effect. Input: **0...999.999**
	Q465 Starting path? Length of the path in the direction of the pitch at which the feed axes are accelerated to the required speed. The approach path is outside of the defined thread contour. This value has an incremental effect. Input: **0.1...99.9**
	Q466 Overrun path? Input: **0.1...99.9**
	Q463 Maximum cutting depth? Maximum infeed perpendicular to the thread pitch Input: **0,001...999.999**

Help graphic	Parameter
	Q467 Feed angle? Angle at which the infeed **Q463** occurs. The reference angle is formed by the parallel line to the thread pitch. Input: **0...60**
	Q468 Infeed type (0/1)? Define the type of infeed: **0**: Consistent chip cross section (the infeed becomes less as the depth increases) **1**: Constant plunging depth Input: **0**, **1**
	Q470 Starting angle? Angle of the turning spindle at which the thread is to be started. Input: **0...359999**
	Q475 Number of thread grooves? Number of thread grooves Input: **1...500**
	Q476 Number of air cuts? Number of air cuts without infeed at finished thread depth Input: **0...255**

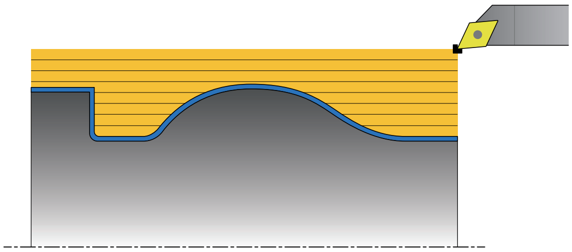
Example

```
11 CYCL DEF 14.0 CONTOUR
12 CYCL DEF 14.1 CONTOUR LABEL2
13 CYCL DEF 830 THREAD CONTOUR-PARALLEL ~
    Q471=+0                ;THREAD POSITION ~
    Q461=+0                ;THREAD ORIENTATION ~
    Q460=+2                ;SAFETY CLEARANCE ~
    Q472=+2                ;THREAD PITCH ~
    Q473=+0                ;DEPTH OF THREAD ~
    Q474=+0                ;THREAD RUN-OUT ~
    Q465=+4                ;STARTING PATH ~
    Q466=+4                ;OVERRUN PATH ~
    Q463=+0.5              ;MAX. CUTTING DEPTH ~
    Q467=+30               ;ANGLE OF INFEED ~
    Q468=+0                ;TYPE OF INFEED ~
    Q470=+0                ;STARTING ANGLE ~
    Q475=+30               ;NUMBER OF STARTS ~
    Q476=+30               ;NUMBER OF AIR CUTS
14 L  X+80  Y+0  Z+2 R0 FMAX M303
15 CYCL CALL
16 M30
17 LBL 2
18 L  X+60  Z+0
19 L  X+70  Z-30
20 RND R60
21 L  Z-45
22 LBL 0
```

14.4.33 Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING (option158)

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



In Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**, the defined contour area is roughed simultaneously in several steps using a movement that includes at least 3 axes (two linear axes and one rotary axis). This allows machining of complex contours with a single tool. During machining, the cycle continuously adjusts the tool angle of inclination based on the following criteria:

- Avoiding collisions between the workpiece, the tool, and the tool carrier
- The tooth does not suffer single-spot wear
- Undercuts are possible

Execution with a FreeTurn tool

You can execute this cycle with FreeTurn tools. This method allows you to perform the most common turning operations with just one tool. Machining times can be reduced through the flexible tool because fewer tool changes occur.

Requirements:

- This function must be enabled by your machine manufacturer.
- You must properly define the tool.
  "Turning operation with FreeTurn tools"

i	The NC program remains unchanged until the FreeTurn cutting edge is called, see "Example: Turning with a FreeTurn tool", Page 833

Roughing cycle run

1 The cycle positions the tool at the cycle start position (tool position when the cycle is called), taking the first tool angle of inclination into account. Then, the tool moves to set-up clearance. If the angle of inclination cannot be achieved at the cycle start position, the control first moves the tool to set-up clearance and from there tilts it using the first tool angle of inclination.
2 The tool moves to the plunging depth **Q519**. The profile infeed may be exceeded for a short time up to the value of **Q463 MAX. CUTTING DEPTH**, e.g. in the case of a corner.
3 The contour is roughed simultaneously using the roughing feed-rate in **Q478**. If you define the plunging feed rate **Q488** in the cycle, it will be effective for the plunging elements. Machining depends on the following input parameters:
   - **Q590**: **MACHINING MODE**
   - **Q591**: **MACHINING SEQUENCE**
   - **Q389**: **UNI.- BIDIRECTIONAL**
4 After each infeed, the control lifts the tool in rapid traverse by the set-up clearance value.
5 The control repeats steps 2 to 4 until the contour has been machined completely.
6 The control retracts the tool at the machining feed rate by the set-up clearance value and then moves it with rapid traverse to the starting position (first in the X axis and then in the Z axis direction)

Notes

NOTICE

Danger of collision!

The control does not perform collision monitoring (DCM). Danger of collision during machining!

- Run a simulation to verify the sequence and the contour
- Verify the NC program by slowly executing it block by block

NOTICE

Danger of collision!

The cycle uses the position of the tool at cycle call as the cycle starting position. Incorrect pre-positioning can cause contour damage. There is a danger of collision!

- Move the tool to a safe position in the X and Z axes.

NOTICE

Danger of collision!

If the contour ends too closely at the fixture, a collision between tool and fixture might occur during machining.

- When clamping, take both the tool angle of inclination and the departure movement into account

NOTICE

Danger of collision!

Collision monitoring only considers the two-dimensional X-Z working plane. The cycle does not check for collisions with an area in the Y coordinate of the cutting edge, tool holder, or tilting body.

- Verify the NC program in **Program Run** in **Single Block**
- Limit the machining area

NOTICE

Danger of collision!

Depending on the geometry of the cutting edge, residual material may be left over. Danger of collision during subsequent machining operations!

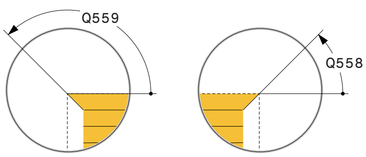
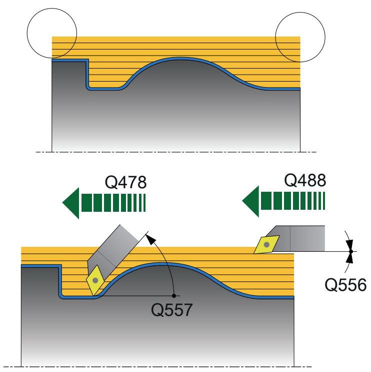
- Run a simulation to verify the sequence and the contour

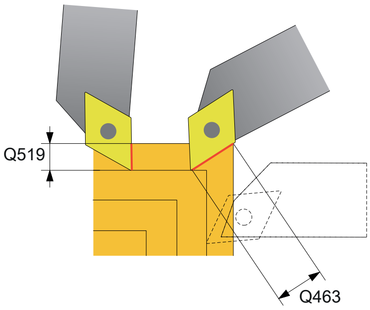
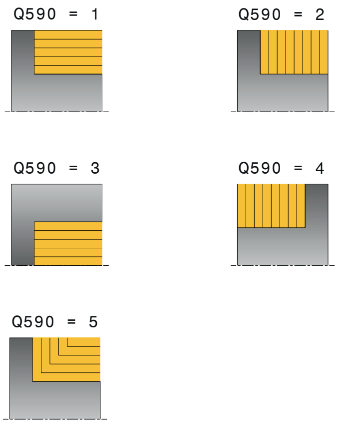
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- If you programmed **M136** before the cycle call, the control interprets the feed rate in millimeters per revolution.
- Software limit switches limit the possible inclination angle **Q556** and **Q557**. If, in **Editor** in the **Simulation** the switch for the software end switches is deactivated, then the simulation may deviate from the later machining operation.
- If it is not possible to machine a particular contour area using this cycle, the control tries to divide the contour area into subareas that can be reached so as to machine them individually.

Notes on programming

- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- Prior to the cycle call, you must program **FUNCTION TCPM**. In **FUNCTION TCMP**, HEIDENHAIN recommends programming the tool reference point **REFPNT TIP-CENTER**.
- The cycle requires a radius compensation (**RL**/**RR**) in its contour description.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- For determining the inclination angle, the cycle requires the definition of a tool holder. For this purpose, assign a tool holder to the tool in the **KINEMATIC** column of the tool table.
- Define a value in **Q463 MAX. CUTTING DEPTH** relative to the cutting edge because, depending on the tool inclination, the infeed from **Q519** may be temporarily exceeded. Use this parameter to limit the extent to which the infeed may be exceeded.

Cycle parameters

Help graphic	Parameter
	Q460 Set-up clearance? Retraction before and after a cut. And distance for the pre-positioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
	**Q558 Extensn. angle at contour start?** Angle in the WPL-CS, by which the cycle extends the contour up to the workpiece blank at the programmed starting point. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	**Q559 Extension angle at contour end?** Angle in WPL CS by which the cycle extends the contour at the programmed end point up to the workpiece blank. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	Q478 Roughing feed rate? Feed rate during roughing in millimeters per minute Input: **0...99999.999** or **FAUTO**
	Q488 Feed rate for plunging Feed rate in millimeters per minute for plunging. This input value is optional. If you do not program the feed rate for plunging, the roughing feed rate **Q478** will apply. Input: **0...99999.999** or **FAUTO**
	Q556 Minimum angle of inclination? Smallest possible permitted angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q557 Maximum angle of inclination? Largest possible angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q567 Finishing allowance of contour? Contour-parallel oversize that will remain after roughing. This value has an incremental effect. Input: **-9...99.999**

Help graphic	Parameter
 	**Q519 Infeed on contour?** Axial, radial and contour-parallel infeed (per cut). Enter a value greater than 0. This value has an incremental effect. Input: **0,001...99.999**
	Q463 Maximum cutting depth? Limit of the maximum infeed relative to the cutting edge. Depending on the tool angle of inclination, the control may temporarily exceed the **Q519 INFEED**, e.g., when machining a corner. Use this optional parameter to limit the extent by which the infeed may be exceeded. If you define the value 0, the maximum infeed is two thirds of the length of the cutting edge. Input: **0...99.999**
 	**Q590 Machining mode (0/1/2/3/4/5)?** Defining the direction of machining: **0**: Automatic; the control automatically combines transverse and longitudinal machining. **1**: Longitudinal turning (outside) **2**: Face turning (front face) **3**: Longitudinal turning (inside) **4**: Face turning (chuck) **5**: Contour-parallel Input: **0**, **1**, **2**, **3**, **4**, **5**
	Q591 Machining sequence (0/1)? Define the machining sequence after which the control executes the contour: **0**: Machining occurs in segments. The sequence is selected in such a way that the center of gravity of the workpiece is shifted towards the chuck as soon as possible. **1**: The workpiece is machined paraxially. The sequence is selected in such a way that the moment of inertia of the workpiece decreases as soon as possible. Input: **0**, **1**
	Q389 Machining strategy (0/1)? Definite the cutting direction: **0**: Unidirectional; every cut is made in the direction of the contour. The direction of the contour depends on **Q499** **1**: Bidirectional; cuts are made against the direction of the contour. The cycle determines the best direction for each following step. Input: **0**, **1**

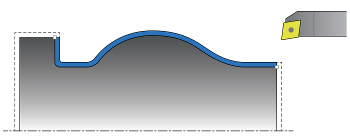
Example

```
11 CYCL DEF 882 SIMULTANEOUS ROUGHING FOR TURNING ~
    Q460=+2                 ;SAFETY CLEARANCE ~
    Q499=+0                 ;REVERSE CONTOUR ~
    Q558=+0                 ;EXT:ANGLE CONT.START ~
    Q559=+90                ;CONTOUR END EXT ANGL ~
    Q478=+0.3               ;ROUGHING FEED RATE ~
    Q488=+0.3               ;PLUNGING FEED RATE ~
    Q556=+0                 ;MIN. INCLINAT. ANGLE ~
    Q557=+90                ;MAX. INCLINAT. ANGLE ~
    Q567=+0.4               ;FINISH. ALLOW. CONT. ~
    Q519=+2                 ;INFEED ~
    Q463=+3                 ;MAX. CUTTING DEPTH ~
    Q590=+0                 ;MACHINING MODE ~
    Q591=+0                 ;MACHINING SEQUENCE ~
    Q389=+1                 ;UNI.- BIDIRECTIONAL
12 L  X+58  Y+0 FMAX M303
13 L  Z+50 FMAX
14 CYCL CALL
```

14.4.34 Cycle 883 TURNING SIMULTANEOUS FINISHING (option 158)

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.
> The cycle is machine-dependent.



You can use this cycle to machine complex contours that are only accessible with different inclinations. When machining with this cycle, the inclination between tool and workpiece changes. This results in machining operations with at least 3 axes (two linear axes and one rotary axis).

The cycle monitors the workpiece contour with respect to the tool and the tool carrier. The cycle avoids unnecessary tilting movements in order to machine optimum surfaces.

If you want to force tilting movements, you can define inclination angles at the beginning and at the end of the contour. Even if simple contours have to be machined, you can use a large area of the indexable insert to achieve longer tool life.

Execution with a FreeTurn tool

You can execute this cycle with FreeTurn tools. This method allows you to perform the most common turning operations with just one tool. Machining times can be reduced through the flexible tool because fewer tool changes occur.

Requirements:

- This function must be enabled by your machine manufacturer.
- You must properly define the tool.

  "Turning operation with FreeTurn tools"

> The NC program remains unchanged until the FreeTurn cutting edge is called, see "Example: Turning with a FreeTurn tool", Page 833

Finishing cycle run

The control uses the tool position as cycle starting point when the cycle is called. If the Z coordinate of the starting point is less than the contour starting point, the control positions the tool in the Z coordinate to set-up clearance and begins the cycle there.

1. The control moves the tool to the set-up clearance **Q460**. The movement is performed at rapid traverse.
2. If programmed, the tool traverses to the inclination angle that was calculated by the control based on the minimum and maximum inclination angles you have defined.
3. The control finishes the contour of the finished part (contour starting point to contour end point) simultaneously at the defined feed rate **Q505**.
4. The control retracts the tool at the defined feed rate to the set-up clearance.
5. The control returns the tool at rapid traverse to the cycle starting point.

Notes

NOTICE
Danger of collision! The control does not perform collision monitoring (DCM). Danger of collision during machining! ▶ Run a simulation to verify the sequence and the contour ▶ Verify the NC program by slowly executing it block by block

NOTICE
Danger of collision! The cycle uses the position of the tool at cycle call as the cycle starting position. Incorrect pre-positioning can cause contour damage. There is a danger of collision! ▶ Move the tool to a safe position in the X and Z axes.

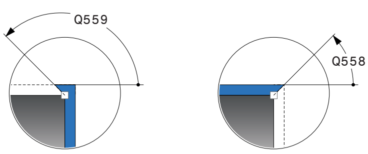
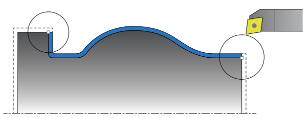
NOTICE
Danger of collision! If the contour ends too closely at the fixture, a collision between tool and fixture might occur during machining. ▶ When clamping, take both the tool angle of inclination and the departure movement into account

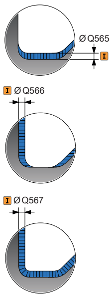
- This cycle can only be executed in the **FUNCTION MODE TURN** machining mode.
- Based on the programmed parameters, the control calculates only **one** collision-free path.
- Software limit switches limit the possible inclination angle **Q556** and **Q557**. If, in **Editor** in the **Simulation** the switch for the software end switches is deactivated, then the simulation may deviate from the later machining operation.
- The cycle calculates a collision-free path. For this purpose, it only uses the 2-D contour of the tool holder without considering the Y axis depth.

Notes on programming

- Before the programming the cycle call, make sure to program Cycle **14 CONTOUR** or **SEL CONTOUR** to be able to define the subprograms.
- Move the tool to a safe position before the cycle call.
- The cycle requires a radius compensation (**RL**/**RR**) in its contour description.
- Prior to the cycle call, you must program **FUNCTION TCPM**. In **FUNCTION TCMP**, HEIDENHAIN recommends programming the tool reference point **REFPNT TIP-CENTER**.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.
- Please note: The smaller the resolution in cycle parameter **Q555** is, the easier will it be to find a solution even in complex situations. The drawback is that the calculation will take more time.
- For determining the inclination angle, the cycle requires the definition of a tool holder. For this purpose, assign a tool holder to the tool in the **KINEMATIC** column of the tool table.
- Please note that cycle parameters **Q565** (Finishing allowance in diameter) and **Q566** (Finishing allowance in Z) cannot be combined with **Q567** (Finishing allowance of contour)!

Cycle parameters

Help graphic	Parameter
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q499 Reverse the contour (0-2)? Define the machining direction of the contour: **0**: Contour is executed in the programmed direction **1**: Contour is executed in the direction opposite to the programmed direction **2**: Contour is executed in the direction opposite to the programmed direction; the position of the tool is also adjusted Input: **0**, **1**, **2**
 	**Q558 Extensn. angle at contour start?** Angle in the WPL-CS, by which the cycle extends the contour up to the workpiece blank at the programmed starting point. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	Q559 Extension angle at contour end? Angle in WPL CS by which the cycle extends the contour at the programmed end point up to the workpiece blank. This angle is used to prevent damage to the workpiece blank. Input: **-180...+180**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q556 Minimum angle of inclination? Smallest possible permitted angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q557 Maximum angle of inclination? Largest possible angle of inclination between the tool and workpiece relative to the Z axis. Input: **-180...+180**
	Q555 Stepping angle for calculation? Cutting width for the calculation of possible solutions Input: **0.5...9.99**

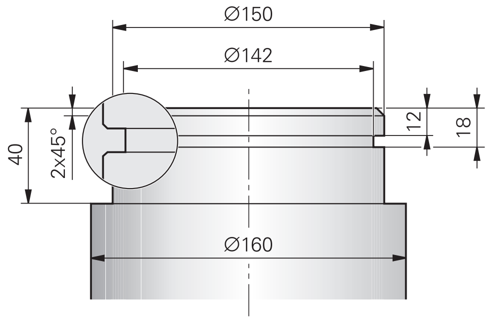
Help graphic	Parameter
	Q537 Inclin. angle (0=N/1=J/2=S/3=E)? Define whether an inclination angle is active: **0**: No inclination angle active **1**: Inclination angle active **2**: Inclination angle at contour start active **3**: Inclination angle at contour end active Input: **0**, **1**, **2**, **3**
	Q538 Inclin. angle at contour start? Inclination angle at the beginning of the programmed contour (WPL-CS) Input: **-180...+180**
	**Q539 Inclinatn. angle at contour end?** Inclination angle at the end of the programmed contour (WPL-CS) Input: **-180...+180**
	Q565 Finishing allowance in diameter Diameter oversize that remains on the contour after finishing. This value has an incremental effect. Input: **-9...99.999**
	Q566 Finishing allowance in Z? Oversize on the defined contour in the axial direction that remains on the contour after finishing. This value has an incremental effect. Input: **-9...99.999**
	Q567 Finishing allowance of contour? Contour-parallel oversize on the defined contour that remains after finishing. This value has an incremental effect. Input: **-9...99.999**

Example

```
11 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING ~
   Q460=+2                  ;SAFETY CLEARANCE ~
   Q499=+0                  ;REVERSE CONTOUR ~
   Q558=+0                  ;EXT:ANGLE CONT.START ~
   Q559=+90                 ;CONTOUR END EXT ANGL ~
   Q505=+0.2                ;FINISHING FEED RATE ~
   Q556=-30                 ;MIN. INCLINAT. ANGLE ~
   Q557=+30                 ;MAX. INCLINAT. ANGLE ~
   Q555=+7                  ;STEPPING ANGLE ~
   Q537=+0                  ;INCID. ANGLE ACTIVE ~
   Q538=+0                  ;INCLIN. ANGLE START ~
   Q539=+0                  ;INCLINATN. ANGLE END ~
   Q565=+0                  ;FINISHING ALLOW. D. ~
   Q566=+0                  ;FINISHING ALLOW. Z ~
   Q567=+0                  ;FINISH. ALLOW. CONT.
12 L  X+58  Y+0 FMAX M303
13 L  Z+50 FMAX
14 CYCL CALL
```

14.4.35 Programming examples

Example: Shoulder with recess



0 BEGIN PGM 9 MM	
1 BLK FORM CYLINDER Z R80 L60	
2 TOOL CALL 301	; tool call
3 M140 MB MAX	; retract the tool
4 FUNCTION MODE TURN	; activate turning mode
5 FUNCTION TURNDATA SPIN VCONST:ON VC:150	; constant cutting speed
6 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+0 ;INCLINED MACHINING ~	
Q531=+0 ;ANGLE OF INCIDENCE ~	
Q532=+750 ;FEED RATE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP	
7 M136	; feed rate in mm/rev.
8 L X+165 Y+0 R0 FMAX	; approach starting point in the plane
9 L Z+2 R0 FMAX M304	; safety clearance, turning spindle on
10 CYCL DEF 812 SHOULDER, LONG. EXT. ~	
Q215=+0 ;MACHINING OPERATION ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q491=+160 ;DIAMETER AT CONTOUR START ~	
Q492=+0 ;CONTOUR START IN Z ~	
Q493=+150 ;DIAMETER AT CONTOUR END ~	
Q494=-40 ;CONTOUR END IN Z ~	
Q495=+0 ;ANGLE OF CIRCUM. SURFACE ~	
Q501=+1 ;TYPE OF STARTING ELEMENT ~	
Q502=+2 ;SIZE OF STARTING ELEMENT ~	
Q500=+1 ;RADIUS OF CONTOUR EDGE ~	
Q496=+0 ;ANGLE OF FACE ~	
Q503=+1 ;TYPE OF END ELEMENT ~	

Q504=+2 ;SIZE OF END ELEMENT ~	
Q463=+2.5 ;MAX. CUTTING DEPTH ~	
Q478=+0.25 ;ROUGHING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q484=+0.2 ;OVERSIZE IN Z ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q506=+0 ;CONTOUR SMOOTHING	
11 CYCL CALL	; cycle call
12 M305	; turning spindle off
13 TOOL CALL 307	; tool call
14 M140 MB MAX	; retract the tool
15 FUNCTION TURNDATA SPIN VCONST:ON VC:100	; constant cutting speed
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+0 ;INCLINED MACHINING ~	
Q531=+0 ;ANGLE OF INCIDENCE ~	
Q532=+750 ;FEED RATE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q535=+0 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP	
17 L X+165 Y+0 R0 FMAX	; approach starting point in the plane
18 L Z+2 R0 FMAX M304	; safety clearance, turning spindle on
19 CYCL DEF 862 EXPND. RECESS, RADL. ~	
Q215=+0 ;MACHINING OPERATION ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q491=+150 ;DIAMETER AT CONTOUR START ~	
Q492=-12 ;CONTOUR START IN Z ~	
Q493=+142 ;DIAMETER AT CONTOUR END ~	
Q494=-18 ;CONTOUR END IN Z ~	
Q495=+0 ;ANGLE OF SIDE ~	
Q501=+1 ;TYPE OF STARTING ELEMENT ~	
Q502=+1 ;SIZE OF STARTING ELEMENT ~	
Q500=+0 ;RADIUS OF CONTOUR EDGE ~	
Q496=+0 ;ANGLE OF SECOND SIDE ~	
Q503=+1 ;TYPE OF END ELEMENT ~	
Q504=+1 ;SIZE OF END ELEMENT ~	
Q478=+0.3 ;ROUGHING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q484=+0.2 ;OVERSIZE IN Z ~	
Q505=+0.15 ;FINISHING FEED RATE ~	
Q463=+0 ;LIMIT TO DEPTH ~	

Q510=+0.8 ;RECESSING OVERLAP ~	
Q511=+80 ;FEED RATE FACTOR ~	
Q462=+0 ;RETRACTION MODE ~	
Q211=+3 ;DWELL TIME IN REVS ~	
Q562=+1 ;MULTIPLE PLUNGING	
20 CYCL CALL M8	; cycle call
21 M305	; turning spindle off
22 M137	; feed rate in mm/minute
23 M140 MB MAX	; retract the tool
24 FUNCTION MODE MILL	; activate milling mode
25 M30	; end of program
26 END PGM 9 MM	

Example: turning, simultaneous finishing

The following NC program illustrates the use of Cycle **883 TURNING SIMULTANEOUS FINISHING**.

Program sequence

- Tool call:Turning tool
- Start turning mode
- Move to safe position
- Call the cycle
- Reset the coordinate system with Cycle **801** and **M145**

0 BEGIN PGM 10 MM	
1 BLK FORM CYLINDER Z D91 L40 DIST+0.5 DI57.5	
2 TOOL CALL 304	; tool call
3 L Z+0 R0 FMAX M91	; retract the tool
4 FUNCTION MODE TURN	; activate turning mode
5 FUNCTION TURNDATA SPIN VCONST:ON VC:200 SMAX800	; constant cutting speed
6 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+1 ;ANGLE OF INCIDENCE ~	
Q532= MAX ;FEED RATE ~	
Q533=+1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP	
7 M145	
8 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT TIP-CENTER	; activate TCPM
9 CYCL DEF 14.0 CONTOUR	
10 CYCL DEF 14.1 CONTOUR LABEL2	
11 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=-90 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q556=-80 ;MIN. INCLINAT. ANGLE ~	
Q557=+60 ;MAX. INCLINAT. ANGLE ~	
Q555=+1 ;STEPPING ANGLE ~	
Q537=+0 ;INCID. ANGLE ACTIVE ~	
Q538=+0 ;INCLIN. ANGLE START ~	
Q539=+50 ;INCLINATN. ANGLE END ~	
Q565=+0 ;FINISHING ALLOW. D. ~	

Q566=+0 ;FINISHING ALLOW. Z ~	
Q567=+0 ;FINISH. ALLOW. CONT.	
12 L X+58 Y+0 R0 FMAX M303	; approach starting point
13 L Z+50 FMAX	; safety clearance
14 CYCL CALL	; cycle call
15 L Z+50 FMAX	
16 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
17 M144	; cancel M145
18 FUNCTION MODE MILL	; activate milling mode
19 M30	; end of program
20 LBL 2	
21 L X+58 Y+0 Z-1.5 RR	
22 L X+61 Z+0	
23 L X+88	
24 L X+90 Z-1	
25 L Z-8	
26 L X+88 Z-10	
27 L Z-15	
28 L X+90 Z-17	
29 L Z-25	
30 RND R0.3	
31 L X+144	
32 LBL 0	
33 END PGM 10 MM	

Example: Turning with a FreeTurn tool

Cycles **882 SIMULTANEOUS ROUGHING FOR TURNING** and **883 TURNING SIMULTANEOUS FINISHING** are used in the following NC program.

Program sequence:

- Activate turning mode
- Call FreeTurn tool with second cutting edge
- Adjust the coordinate system with cycle **800 ADJUST XZ SYSTEM**
- Move to safe position
- Call cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**
- Call FreeTurn tool with second cutting edge
- Move to safe position
- Call cycle **882 SIMULTANEOUS ROUGHING FOR TURNING**
- Move to safe position
- Call cycle **883 TURNING SIMULTANEOUS FINISHING**
- Reset active transformation with the PC program **RESET.h**

0 BEGIN PGM FREETURN MM	
1 FUNCTION MODE TURN "AC_TURN"	; activate turning mode
2 PRESET SELECT #16	
3 BLK FORM CYLINDER Z D100 L101 DIST+1	
4 FUNCTION TURNDATA BLANK LBL 1	; activate blank form update
5 TOOL CALL 145.0	; call FreeTurn tool with first edge
6 M136	
7 FUNCTION TURNDATA SPIN VCONST:ON VC:250	; constant cutting speed
8 L Z+50 R0 FMAX M303	
9 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+90 ;ANGLE OF INCIDENCE ~	
Q532= MAX ;FEED RATE ~	
Q533=-1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	
10 CYCL DEF 14.0 CONTOUR	
11 CYCL DEF 14.1 KONTURLABEL2	
12 CYCL DEF 882 SIMULTANEOUS ROUGHING FOR TURNING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=+0 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q478=+0.3 ;ROUGHING FEED RATE ~	
Q488=+0.3 ;PLUNGING FEED RATE ~	

Q556=+30 ;MIN. INCLINAT. ANGLE ~	
Q557=+160 ;MAX. INCLINAT. ANGLE ~	
Q567=+0.3 ;FINISH. ALLOW. CONT. ~	
Q519=+2 ;INFEED ~	
Q463=+2 ;MAX. CUTTING DEPTH ~	
Q590=+5 ;MACHINING MODE ~	
Q591=+1 ;MACHINING SEQUENCE ~	
Q389=+0 ;UNI.- BIDIRECTIONAL	
13 L X+105 Y+0 R0 FMAX	
14 L Z+2 R0 FMAX M99	
15 TOOL CALL 145.1	; call FreeTurn tool with second cutting edge
16 CYCL DEF 800 ADJUST XZ SYSTEM ~	
Q497=+0 ;PRECESSION ANGLE ~	
Q498=+0 ;REVERSE TOOL ~	
Q530=+2 ;INCLINED MACHINING ~	
Q531=+90 ;ANGLE OF INCIDENCE ~	
Q532= MAX ;FEED RATE ~	
Q533=-1 ;PREFERRED DIRECTION ~	
Q535=+3 ;ECCENTRIC TURNING ~	
Q536=+0 ;ECCENTRIC W/O STOP ~	
Q599=+0 ;RETRACT	
17 Q519 = 1	; reduce infeed to 1
18 L X+105 Y+0 R0 FMAX	; approach starting point
19 L Z+2 R0 FMAX M99	; call cycle
20 CYCL DEF 883 TURNING SIMULTANEOUS FINISHING ~	
Q460=+2 ;SAFETY CLEARANCE ~	
Q499=+0 ;REVERSE CONTOUR ~	
Q558=+0 ;EXT:ANGLE CONT.START ~	
Q559=+90 ;CONTOUR END EXT ANGL ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q556=+30 ;MIN. INCLINAT. ANGLE ~	
Q557=+160 ;MAX. INCLINAT. ANGLE ~	
Q555=+5 ;STEPPING ANGLE ~	
Q537=+0 ;INCID. ANGLE ACTIVE ~	
Q538=+90 ;INCLIN. ANGLE START ~	
Q539=+0 ;INCLINATN. ANGLE END ~	
Q565=+0 ;FINISHING ALLOW. D. ~	
Q566=+0 ;FINISHING ALLOW. Z ~	
Q567=+0 ;FINISH. ALLOW. CONT.	
21 L X+105 Y+0 R0 FMAX	; approach starting point
22 L Z+2 R0 FMAX M99	; call cycle
23 CALL PGM RESET.H	; call **RESET** program

24 M30	; end of program
25 LBL 1	; define **LBL 1**
26 L X+100 Z+1	
27 L X+0	
28 L Z-60	
29 L X+100	
30 L Z+1	
31 LBL 0	
32 LBL 2	; define **LBL 2**
33 L Z+1 X+60 RR	
34 L Z+0	
35 L Z-2 X+70	
36 RND R2	
37 L X+80	
38 RND R2	
39 L Z+0 X+98	
40 RND R2	
41 L Z-10	
42 RND R2	
43 L Z-8 X+89	
44 RND R2	
45 L Z-15 X+60	
46 RND R2	
47 L Z-55	
48 RND R2	
49 L Z-50 X+98	
50 RND R2	
51 L Z-60	
52 LBL 0	
53 END PGM FREETURN MM	

14.5 Cycles for grinding

14.5.1 Overview

Reciprocating stroke

Cycle	Call	Further information
1000 DEFINE RECIP. STROKE (option 156) ■ Define the reciprocating stroke and start it, if applicable	**DEF**-active	Page 838
1001 START RECIP. STROKE (option 156) ■ Start reciprocating stroke	**DEF**-active	Page 840
1002 STOP RECIP. STROKE (option 156) ■ Stop the reciprocating stroke and clear it, if applicable	**DEF**-active	Page 842

Dressing cycles

Cycle	Call	Further information
1010 DRESSING DIAMETER (option 156) ■ Dressing a grinding wheel diameter	**DEF**-active	Page 845
1015 PROFILE DRESSING (option 156) ■ Dressing a defined grinding wheel profile	**DEF**-active	Page 848
1016 DRESSING OF CUP WHEEL (option 156) ■ Dressing a cup wheel	**DEF**-active	Page 853
1017 DRESSING WITH DRESSING ROLL (option 156) ■ Dressing with a dressing roll ■ Reciprocating strokes ■ Oscillating ■ Fine oscillating	**DEF**-active	Page 858
1018 RECESSING WITH DRESSING ROLL (option 156) ■ Dressing with a dressing roll ■ Recessing ■ Multi-recessing	**DEF**-active	Page 864

Contour grinding cycles

Cycle	Call	Further information
1021 CYLINDER, SLOW-STROKE GRINDING (option 156) ■ Grinding of cylindrical internal or external contours ■ Multiple circular paths during a reciprocating stroke	**CALL**-active	Page 870
1022 CYLINDER, FAST-STROKE GRINDING (option 156) ■ Grinding of cylindrical internal or external contours ■ Grinding with circular and helix paths, movement superimposed w/ reciprocating stroke, as required	**CALL**-active	Page 877
1025 GRINDING CONTOUR (option 156) ■ Grinding open and closed contours	**CALL**-active	Page 884

Special cycles

Cycle	Call	Further information
1030 **ACTIVATE WHEEL EDGE** (option 156) ■ Activating the desired wheel edge	**DEF-**active	Page 887
1032 **GRINDING WHL LENGTH COMPENSATION** (option 156) ■ Compensation of the length in absolute or incremental values	**DEF-**active	Page 889
1033 **GRINDING WHL RADIUS COMPENSATION** (option 156) ■ Compensation of the radius in absolute or incremental values	**DEF-**active	Page 891

14.5.2 General information on jig grinding

General information on jig grinding

Jig grinding means grinding of a 2-D contour. There is not much of a difference between jig grinding and milling. Instead of a milling cutter, a grinding tool is used, such as a grinding pin. Machining is performed in milling mode, i.e. with **FUNCTION MODE MILL**.

Grinding cycles provide special movements for the grinding tool. A stroke or oscillating movement, the so-called reciprocating stroke, is superimposed with the movement in the working plane.

Outline: Grinding with a reciprocating stroke

```
0 BEGIN PGM GRIND MM
1 FUNCTION MODE MILL
2 TOOL CALL "GRIND_1" Z S20000
3 CYCL DEF 1000 DEFINE RECIP. STROKE
...
4 CYCL DEF 1001 START RECIP. STROKE
...
5 CYCL DEF 14 CONTOUR
...
6 CYCL DEF 1025 GRINDING CONTOUR
...
7 CYCL CALL
8 CYCL DEF 1002 STOP RECIP. STROKE
...
9 END PGM GRIND MM
```

14.5.3 Cycle 1000 DEFINE RECIP. STROKE (option 156)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1000 DEFINE RECIP. STROKE** to define a reciprocating stroke in the tool axis and start reciprocating. This movement is executed as a superimposed movement. Thus, it is possible to execute any positioning block in parallel to the reciprocating stroke, even in the axis that is reciprocating. Once you started the reciprocating stroke, you can call a contour and start grinding.

- If you set **Q1004** to **0**, no reciprocating stroke takes place. In this case, you only define the cycle. If required, call Cycle **1001 START RECIP. STROKE** later to start the reciprocating stroke
- If you set **Q1004** to **1**, the reciprocating stroke starts at the current position. Depending on the setting in **Q1002**, the control will start reciprocating the tool in the positive or negative direction first. This reciprocation movement will be superimposed on the programmed movements (X, Y, Z)

The following cycles can be called in combination with the reciprocating stroke:

- Cycle **24 SIDE FINISHING**
- Cycle **25 CONTOUR TRAIN**
- Cycles **25x POCKETS/STUDS/SLOTS**
- Cycle **276 THREE-D CONT. TRAIN**
- Cycle **274 OCM FINISHING SIDE**
- Cycle **1025 GRINDING CONTOUR**

- The control does not support mid-program startup while the reciprocating stroke is active.
- As long as the reciprocating stroke is active in the started NC program, you cannot switch to **Manual operationMDI** application.

Notes

Refer to your machine manual!

The overrides for the reciprocation movements can be changed by the machine tool builder.

NOTICE

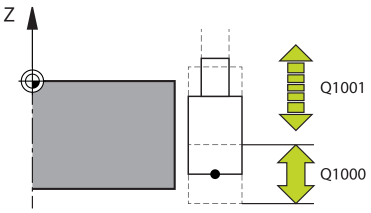
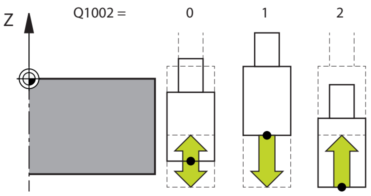
Danger of collision!

Collision monitoring (DCM) is not active during reciprocation movements. This means that movements that might cause collisions will not be prevented. There is a danger of collision!

- Verify the NC program by carefully executing it block by block

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1000** is DEF-active.
- The simulation of the superimposed movement can be seen in **Program Run** mode and in **Single Block** mode.
- Stop the reciprocating movement when you no longer need it. To do so, use **M30** or Cycle **1002 STOP RECIP. STROKE**. **STOP** or **M0** will not stop the reciprocating stroke.
- Reciprocating strokes can also be started in a tilted working plane. While the reciprocating stroke is active, however, you cannot change the orientation of the plane.
- You can also use a milling cutter with the superimposed reciprocating movement.

Cycle parameters

Help graphic	Parameter
	**Q1000 Length of reciprocating stroke?** Length of the reciprocating movement, parallel to the active tool axis Input: **0...9999.9999**
	Q1001 Feed rate for reciprocation? Speed of the reciprocating stroke in mm/min Input: **0...999999**
	**Q1002 Type of reciprocation?** Definition of the start position. The direction of the first reciprocating stroke arises from this. **0**: The current position is the middle of the stroke. The control first offsets the grinding tool by half the stroke in the negative direction and then continues the reciprocating movement in the positive direction **-1**: The current position is the upper limit of the stroke. During the first stroke, the control offsets the grinding tool in the negative direction. **+1**: The current position is the lower limit of the stroke. For the first stroke, the control offsets the grinding tool in the positive direction Input: **-1**, **0**, **+1**
	Q1004 Start reciprocating stroke? Definition of the effect of this cycle: **0**: The reciprocating stroke is merely defined and may be started at a later time **+1**: The reciprocating stroke is defined and started at the current position Input: **0**, **1**

Example

```
11 CYCL DEF 1000 DEFINE RECIP. STROKE ~
    Q1000=+0              ;RECIPROCATING STROKE ~
    Q1001=+999            ;RECIP. FEED RATE ~
    Q1002=+1              ;RECIPROCATION TYPE ~
    Q1004=+0              ;START RECIP. STROKE
```

14.5.4 Cycle 1001 START RECIP. STROKE (option 156)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Cycle **1001 START RECIP. STROKE** starts a previously defined or stropped reciprocation movement. In an ongoing movement, this cycle has no effect.

Notes

	Refer to your machine manual! The overrides for the reciprocation movements can be changed by the machine tool builder.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1001** is DEF-active.
- If you did not define a reciprocating stroke with Cycle **1000 DEFINE RECIP. STROKE**, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	Cycle **1001** does not have a cycle parameter. Close cycle input with the **END** key.

Example

```
11 CYCL DEF 1001 START RECIP. STROKE
```

14.5.5 Cycle 1002 STOP RECIP. STROKE (option 156)

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

Cycle **1002 STOP RECIP. STROKE** stops the reciprocation movement. Depending on the setting in **Q1010**, the tool will stop immediately or traverse to its starting position.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **1002** is DEF-active.

Note on programming

- Stopping the movement at the current position (**Q1010**=1) is allowed only if you simultaneously clear the definition of the reciprocating stroke (**Q1005**=1).

Cycle parameters

Help graphic	Parameter
	Q1005 Clear reciprocating stroke? Definition of the effect of this cycle: **0**: The reciprocating stroke is merely stopped and may be started again at a later time **+1**: The reciprocating stroke is stopped, and the definition of the reciprocating stroke from cycle **1000** is cleared Input: **0**, **1**
	Q1010 Stop reciproc. immediately (1)? Definition of the stopping position of the grinding tool: **0**: The stopping position is the same as the starting position **+1**: The stopping position is the same as the current position Input: **0**, **1**

Example

```
11 CYCL DEF 1002 STOP RECIP. STROKE ~
    Q1005=+0              ;CLEAR RECIP. STROKE ~
    Q1010=+0              ;RECIP.STROKE STOPPOS
```

14.5.6 General information on the dressing cycles

Fundamentals

Refer to your machine manual.

For dressing operations, the machine must be prepared accordingly by the machine manufacturer. The machine manufacturer may provide his own cycles.

The term “dressing” refers to the sharpening or trueing up of a grinding tool inside the machine. During dressing, the dresser machines the grinding wheel. Thus, in dressing, the grinding tool is the workpiece.

The dresser removes material and thereby changes the dimensions of the grinding wheel. Dressing the diameter, for example, causes the radius of the grinding wheel to become smaller.

The following dressing cycles are available:

- **1010 DRESSING DIAMETER**, Page 845
- **1015 PROFILE DRESSING**, Page 848
- **1016 DRESSING OF CUP WHEEL**, Page 853
- **1017 DRESSING WITH DRESSING ROLL**, Page 858
- **1018 RECESSING WITH DRESSING ROLL**, Page 864

In dressing, the workpiece datum is located on an edge of the grinding wheel. Select the respective edge using Cycle **1030 ACTIVATE WHEEL EDGE**.

Identify dressing operations in your NC program with **FUNCTION DRESS BEGIN / END**. When you activate **FUNCTION DRESS BEGIN**, the grinding wheel is redefined as the workpiece and the dressing tool as the tool. This might result in the axes moving in the opposite direction. When you terminate the dressing mode with **FUNCTION DRESS END**, the grinding wheel is redefined as the tool.

Further information: "Dressing", Page 227

Structure of an NC program for dressing:

- Activate milling mode
- Call grinding wheel
- Move the tool to be dressed to a position near the dressing tool
- Activate dressing mode; select the kinematic model if necessary
- Activate wheel edge
- Call dressing tool; no mechanical tool change
- Call the cycle for dressing the diameter
- Deactivate dressing mode

```
0 BEGIN PGM GRIND MM
1 FUNCTION MODE MILL
2 TOOL CALL "GRIND_1" Z S20000
3 L X... Y... Z...
4 FUNCTION DRESS BEGIN
5 CYCL DEF 1030 ACTIVATE WHEEL EDGE
...
6 TOOL CALL "DRESS_1"
7 CYCL DEF 1010 DRESSING DIAMETER
...
8 FUNCTION DRESS END
9 END PGM GRIND MM
```

- The control does not support mid-program startup while dressing is active. If you jump to the first NC block after dressing using mid-program startup, the control will move the tool to the last position approached during dressing.

Notes

- If you interrupt a dressing infeed movement, the last infeed will not be considered. If applicable, the dressing tool executes the first infeed or part of it without removing material if the dressing cycle is called again.
- Not all grinding tools require dressing. Comply with the information provided by your tool manufacturer.
- Please note that the switchover to dressing mode might have been programmed into the cycle sequence already by the machine manufacturer.

  Further information: "Dressing", Page 227

14.5.7 Cycle 1010 DRESSING DIAMETER (option 156)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Cycle **1010 DRESSING DIAMETER** allows you to dress the outside diameter of your grinding wheel. Depending on the strategy, the control causes movements based on the wheel geometry. If the dressing strategy in **Q1016** was set to 1 or 2, the path of the tool to the starting point is not along the grinding wheel, but via a retract path. The control does not apply tool radius compensation in the dressing cycle.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	1, 3, 5, 7	not supported

If you work with the dressing roll tool type, then only the grinding pin is permitted.

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 887

Notes

NOTICE
Danger of collision! When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining! ▶ Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode ▶ Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool ▶ Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine tool builder ▶ In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes ▶ If necessary, program a kinematic switch-over

NOTICE
Danger of collision! The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! ▶ Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool ▶ Make sure there is no risk of collision ▶ Verify the NC program by slowly executing it block by block

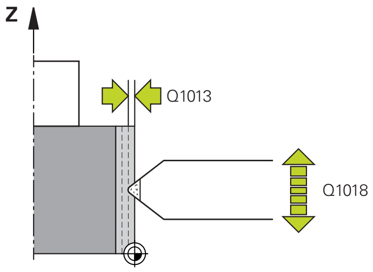
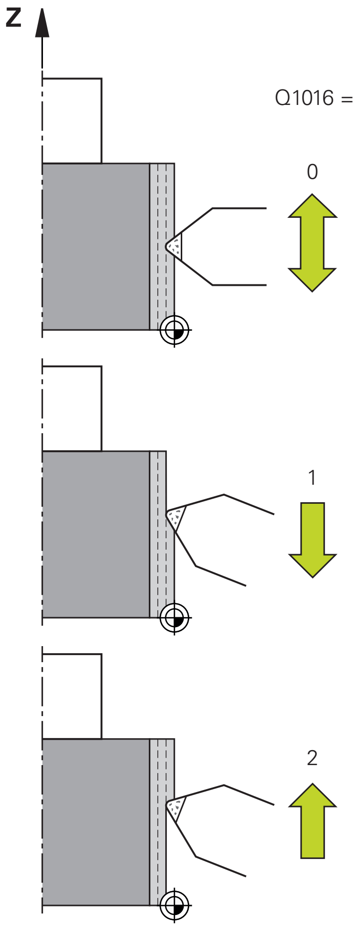
- Cycle **1010** is DEF-active.
- No coordinate transformations are allowed in dressing mode.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, the control executes the dressing procedure only after reaching the defined counter in the tool table. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- The cycle supports dressing with a dressing role.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

  Further information: "Dressing", Page 227

Information about dressing with a dressing role

- For the dressing tool, you must define the dressing role **TYPE**.
- For the dressing role, you must define a width: **CUTWIDTH**. The control takes the width into account during the dressing process.
- For dressing with a dressing role, only the dressing strategy **Q1016=0** is allowed.

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	**Q1016 Dressing strategy (0-2)?** Definition of the traversing movement during dressing: **0**: Reciprocating; dressing occurs in both directions **1**: Pulling; dressing occurs along the grinding wheel solely towards the active wheel edge **2**: Pushing; dressing occurs along the grinding wheel solely away from the active wheel edge Input: **0**, **1**, **2**
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**

Help graphic	Parameter
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1010 DRESSING DIAMETER ~
    Q1013=+0            ;DRESSING AMOUNT ~
    Q1018=+100          ;DRESSING FEED RATE ~
    Q1016=+1            ;DRESSING STRATEGY ~
    Q1019=+1            ;NUMBER INFEEDS ~
    Q1020=+0            ;IDLE STROKES ~
    Q1022=+0            ;COUNTER FOR DRESSING ~
    Q330=-1             ;TOOL ~
    Q1011=+0            ;FACTOR VC
```

14.5.8 Cycle 1015 PROFILE DRESSING (option 156)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1015 PROFILE DRESSING** to dress a defined profile of your grinding wheel. The profile must be defined in a separate NC program. This cycle is based on the grinding pin tool type. The starting point and end point of the profile must be identical (closed path) and are located at a corresponding position on the selected wheel edge. Define the return path to the starting point in your profile program. You must program the NC program in the ZX plane. Depending on the profile program, the control either does or does not use tool radius compensation. The activated wheel edge is used as the preset.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	not supported	not supported

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 887

Cycle sequence

1 The control positions the dressing tool at the starting position with **FMAX**. The distance of the starting position from the datum is equal to the retraction values of the grinding wheel. The retraction values are relative to the active grinding edge.
2 The control offsets the datum to the extent of the dressing value and executes the profile program. This process repeats itself depending on the definition of **NUMBER INFEEDS Q1019**.
3 The control executes the profile program to the extent of the dressing value. If have programmed **NUMBER INFEEDS Q1019**, the infeeds repeat themselves. For every infeed, the dressing tool moves to the extent of the dressing value **Q1013**.
4 The profile program is repeated without infeed in accordance with **IDLE STROKES Q1020**.
5 The motion ends in the starting position.

- The datum of the workpiece system lies on the active wheel edge.

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine tool builder
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE
Danger of collision! The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement! ▶ Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool ▶ Make sure there is no risk of collision ▶ Verify the NC program by slowly executing it block by block

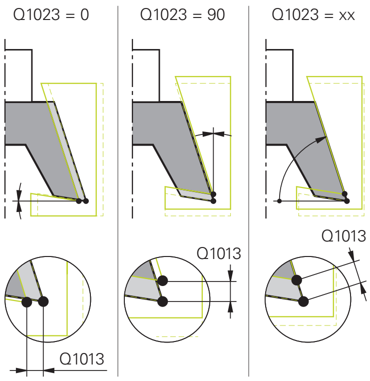
- Cycle **1015** is DEF-active.
- No coordinate transformations are allowed in dressing mode.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, the control executes the dressing procedure only after reaching the defined counter in the tool table. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

  Further information: "Dressing", Page 227

Note on programming

- The angle of infeed must be selected in a way that the programmed profile always remains within the grinding wheel edge. If this condition is not met, then the dimensional accuracy of the grinding wheel is lost.

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1023 Infeed angle of profile program? Angle at which the profile of the program is moved into the grinding wheel. **0**: Infeed only at the diameter in the X axis of the dressing kinematic model **+90**: Infeed only in the Z axis of the dressing kinematic model Input: **0...90**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	Q1000 Name of the profile program? Enter the path and name of the NC program that will be used for the profile of the grinding wheel during the dressing process. Alternatively, select the profile program via name option in the action bar. Input: Max. **255** characters
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**

Help graphic	Parameter
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1015 PROFILE DRESSING ~
   Q1013=+0                 ;DRESSING AMOUNT ~
   Q1023=+0                 ;ANGLE OF INFEED ~
   Q1018=+100               ;DRESSING FEED RATE ~
   QS1000=""                ;PROFILE PROGRAM ~
   Q1019=+1                 ;NUMBER INFEEDS ~
   Q1020=+0                 ;IDLE STROKES ~
   Q1022=+0                 ;COUNTER FOR DRESSING ~
   Q330=-1                  ;TOOL ~
   Q1011=+0                 ;FACTOR VC
```

14.5.9 Cycle 1016 DRESSING OF CUP WHEEL (option 156)

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1016 DRESSING OF CUP WHEEL** to dress the front face of a cup wheel. The activated wheel edge is used as the reference.

Depending on the strategy, the control causes movements based on the wheel geometry. If the dressing strategy in **Q1016** was set to **1** or **2**, the return of the tool to the starting point is not along the grinding wheel, but via a retract path.

If the Pull-and-Push strategy has been selected in dressing mode, the control will apply radius compensation. If the Reciprocating strategy has been selected in dressing mode, the control will not apply radius compensation.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
not supported	not supported	2, 6

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 887

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine tool builder
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement!

- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

NOTICE

Danger of collision!

The angle of inclination between the dressing tool and the cup wheel will not be monitored! There is a danger of collision!

- Make sure to program a dressing tool clearance angle greater than or equal to 0° relative to the front face of the cup wheel
- Verify the NC program by carefully executing it block by block

- Cycle **1016** is DEF-active.
- No coordinate transformations are allowed in dressing mode.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, the control executes the dressing procedure only after reaching the defined counter in the tool table. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.
- The control saves the counter in the tool table. Its effect is global.

  Further information: "Tool data for grinding tools (option 156)", Page 263
- To enable dressing of the entire cutting edge, it is extended by twice the cutting-edge radius (2 x **RS**) of the dressing tool. Here, the minimum permissible radius (**R_MIN**) of the grinding wheel must not be undershot, otherwise the control interrupts the operation with an error message.
- In this cycle, the radius of the tool shank is not monitored.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

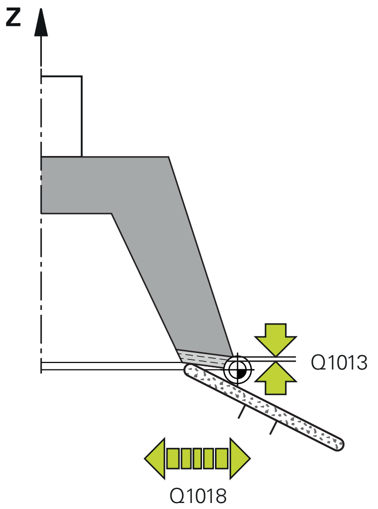
  Further information: "Simplified dressing with a macro", Page 228

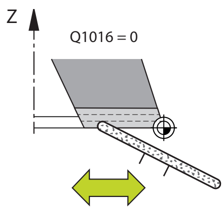
Notes on programming

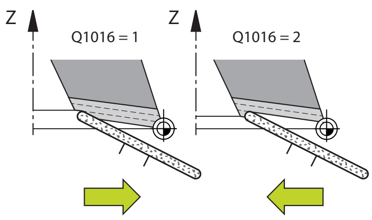
- This cycle is permitted only for use with the cup wheel tool type. If you defined a different tool type, the control will display an error message.
- The strategy in **Q1016** = 0 (Reciprocating) is only possible for a straight front face angle (**HWA** = 0).

Cycle parameters

Help graphic







Parameter
Q1013 Dressing amount? Value used by the control for the dressing infeed. Input: **0...9.9999**
Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
Q1016 Dressing strategy (0-2)? Definition of the traversing movement during dressing: **0**: Reciprocating; dressing occurs in both directions **1**: Pulling; dressing occurs along the grinding wheel solely towards the active wheel edge **2**: Pushing; dressing occurs along the grinding wheel solely away from the active wheel edge Input: **0**, **1**, **2**
Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**

Help graphic	Parameter
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1016 DRESSING OF CUP WHEEL ~
    Q1013=+0                ;DRESSING AMOUNT ~
    Q1018=+100              ;DRESSING FEED RATE ~
    Q1016=+1                ;DRESSING STRATEGY ~
    Q1019=+1                ;NUMBER INFEEDS ~
    Q1020=+0                ;IDLE STROKES ~
    Q1022=+0                ;COUNTER FOR DRESSING ~
    Q330=-1                 ;TOOL ~
    Q1011=+0                ;FACTOR VC
```

14.5.10 Cycle 1017 DRESSING WITH DRESSING ROLL (option 156)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

With cycle **1017 DRESSING WITH DRESSING ROLL**, you can dress the outside diameter of a grinding wheel with a dressing role. Depending on the dressing strategy, the control performs the appropriate movements in accordance with the wheel geometry.

The cycle offers the following dressing strategies:

- Reciprocating: lateral infeed at the reversal points of the reciprocating stroke
- Oscillating: interpolating infeed during a reciprocating stroke
- Fine Oscillating: interpolating infeed during a reciprocating stroke. After each interpolating infeed, a Z movement is performed without infeed in the dressing kinematic model.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	not supported	not supported

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 887

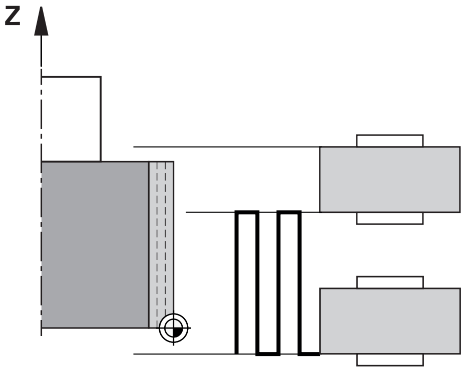
Cycle sequence

1. The control positions the dressing tool at the starting position with **FMAX**.
2. If you have defined a pre-position in **Q1025 PRE-POSITION**, the control approaches the position at **Q253 F PRE-POSITIONING**.
3. The control infeeds based on the dressing strategy.
   Further information: "Dressing strategies", Page 859
4. After defining **IDLE STROKES** in **Q1020**, the control performs them after the last infeed.
5. The control moves to the starting position with **FMAX**.

Dressing strategies

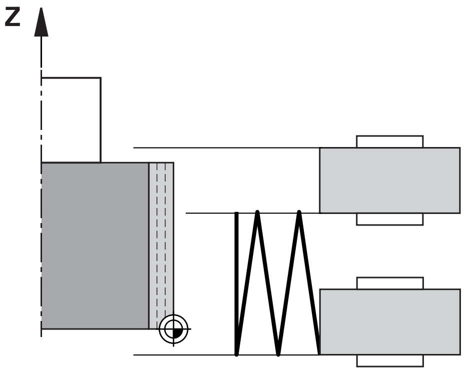
> Depending on **Q1026 WEAR FACTOR**, the control divides the dressing value between the grinding wheel and the dressing roll.

Reciprocating (**Q1024=0**)



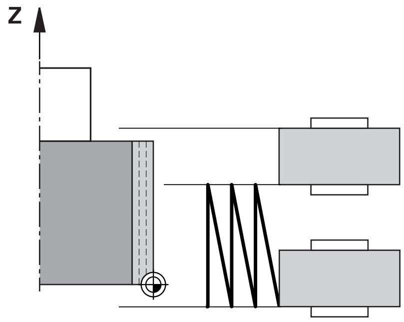
1 The dressing roll approaches the grinding wheel at the **DRESSING FEED RATE Q1018**.
2 The **DRESSING AMOUNT Q1013** is infed on the diameter at the **DRESSING FEED RATE Q1018**.
3 The control moves the dressing tool along the grinding wheel to the next reversal point of the reciprocating movement.
4 If other dressing infeeding is required, the control repeats processes 1 to 2 until the dressing process is complete.

Oscillating (**Q1024=1**)



1 The dressing roll approaches the grinding wheel at the **DRESSING FEED RATE Q1018**.
2 The control infeeds the **DRESSING AMOUNT Q1013** on the diameter. Infeeding is performed with interpolation at the dressing feed rate **Q1018** with the reciprocating stroke up to the next reversal point.
3 If there are more dressing infeed runs, then processes 1 to 2 are repeated until the dressing process is complete.
4 The control then retracts the tool without infeed in the Z axis of the dressing kinematic model to the other reversal point of the reciprocating movement.

Fine oscillating (**Q1024=2**)



1 The dressing roll approaches the grinding wheel at the **DRESSING FEED RATE Q1018**.
2 The control infeeds the **DRESSING AMOUNT Q1013** on the diameter. Infeeding is performed with interpolation at the dressing feed rate **Q1018** with the reciprocating stroke up to the next reversal point.
3 The control then retracts the tool to the other reversal point of the reciprocating movement without an infeed cut.
4 If there is more infeeding, then processes 1 to 3 are repeated until the dressing procedure is complete.

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine tool builder
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

NOTICE

Danger of collision!

The dressing cycles position the dressing tool at the programmed grinding wheel edge. Positioning occurs simultaneously in two axes of the working plane. The control does not perform collision checking during this movement!

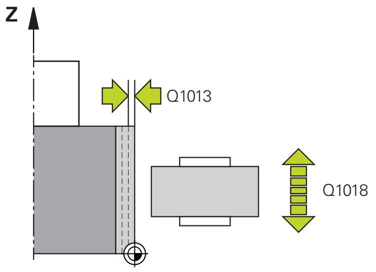
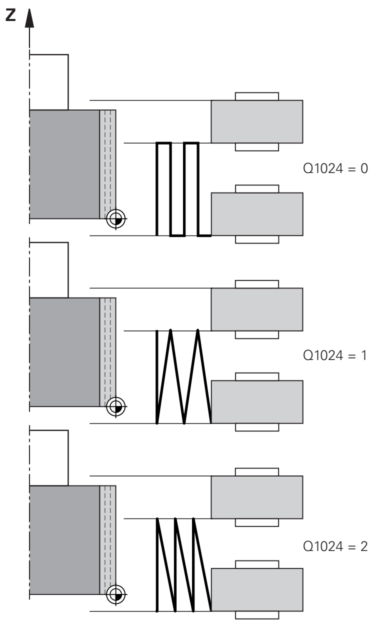
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Make sure there is no risk of collision
- Verify the NC program by slowly executing it block by block

- Cycle **1017** is DEF-active.
- No coordinate conversion cycles are permitted in dressing mode. The control displays an error message.
- The control does not graphically depict the dressing operation.
- If you program a **COUNTER FOR DRESSING Q1022**, then the control performs the dressing process only after reaching the defined counter from the tool management function. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.

  Further information: "Dressing tool table tooldress.drs (option 156)", Page 1836
- At the end of each infeed run, the control updates the tool data for the grinding tool and dressing tool.
- For the reversal points of the reciprocating movement, the control takes into account the retraction values **AA** and **AI** from the tool management function. The width of the dressing roll must be less than the width of the dressing wheel, including the retraction values.
- The control does not apply tool radius compensation in the dressing cycle.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

  Further information: "Simplified dressing with a macro", Page 228

Cycle parameters

Help graphic	Parameter
	**Q1013 Dressing amount?** Value used by the control for the dressing infeed. Input: **0...9.9999**
	Q1018 Feed rate for dressing? Feed rate during the dressing procedure Input: **0...99999**
	**Q1024 Dressing strategy (0-2)?** Strategy during dressing with a dressing roll; **0**: Reciprocating; infeeding to the reversal points of the reciprocating motion. After the infeeding runs, the control executes a movement just in the Z axis within the dressing kinematic model. **1**: Oscillating; interpolated infeed during a reciprocating movement **2**: Fine oscillating; interpolated during a reciprocating movement. After every interpolated infeed run, the control executes a movement solely in the Z axis in the dressing kinematic model. Input: **0**, **1**, **2**
	Q1019 Number of dressing infeeds? Number of infeeds of the dressing process Input: **1...999**
	Q1020 Number of idle strokes? Number of times the dressing tool moves along the grinding wheel without removing material after the most recent infeed. Input: **0...99**
	Q1025 Distance for pre-positioning? Distance between the grinding wheel and the dressing role during pre-positioning Input: **0...9.9999**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool in mm/min. while approaching the pre-position Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q1026 Wear on dressing tool? Factor of the dressing value in order to define the wear on the dressing roll: **0**: The full dressing value is removed on the grinding wheel. **>0**: The factor is multiplied by the dressing value. The control takes the calculated value into account and assumes that this value will be lost during dressing due to wear on the dressing roll. The remaining dressing value is dressed on the grinding wheel. Input: **0...+0.99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1017 DRESSING WITH DRESSING ROLL ~
    Q1013=+0              ;DRESSING AMOUNT ~
    Q1018=+100            ;DRESSING FEED RATE ~
    Q1024=+0              ;DRESSING STRATEGY ~
    Q1019=+1              ;NUMBER INFEEDS ~
    Q1020=+0              ;IDLE STROKES ~
    Q1025=+5              ;PRE-POSITION DIST. ~
    Q253=+1000            ;F PRE-POSITIONING ~
    Q1026=+0              ;WEAR FACTOR ~
    Q1022=+2              ;COUNTER FOR DRESSING ~
    Q330=-1               ;TOOL ~
    Q1011=+0              ;FACTOR VC
```

14.5.11 Cycle 1018 RECESSING WITH DRESSING ROLL (option 156)

Application

> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.

With cycle **1018 RECESSING WITH DRESSING ROLL**, you can dress the outside diameter of a grinding wheel via recessing with dressing role. Depending on the dressing strategy, the control executes one or more recessing movements.

The cycle offers the following dressing strategies:

- **Recessing:** This strategy performs only linear recessing movements. The width of the dressing roll is larger than the dressing wheel width.
- **Multiple recessing:** This strategy executes linear recessing movements. At the end of the infeed run, the control moves the dressing tool in the Z axis of the dressing kinematic model and infeeds again.

This cycle supports the following wheel edges:

Grinding pin	Special grinding pin	Cup wheel
1, 2, 5, 6	not supported	not supported

Further information: "Cycle 1030 ACTIVATE WHEEL EDGE (option 156)", Page 887

Cycle sequence

Recessing

1 The control positions the dressing roll at the starting position with **FMAX**. At the starting position, the middle of the dressing roll matches the middle of the grinding wheel edge. If **CENTER OFFSET Q1028** is programmed, then the control takes this into account when approaching the starting position.
2 The dressing role approaches the **PRE-POSITION DIST. Q1025** with the feed rate **Q253 F PRE-POSITIONING**.
3 The dressing roll recesses into the grinding wheel with the **DRESSING FEED RATE Q1018** by the **DRESSING AMOUNT Q1013**.
4 If a **DWELL TIME IN REVS Q211** is defined, the control waits the defined amount of time.
5 The control retracts the dressing role with **F PRE-POSITIONING Q253** to the **PRE-POSITION DIST. Q1025**.
6 The control moves to the starting position with **FMAX**.

Multiple recessing

1 The control positions the dressing roll at the starting position with **FMAX**.
2 The dressing role approaches the **PRE-POSITION DIST.VORPOSITION Q1025** at the feed rate **Q253F PRE-POSITIONING**.
3 The dressing roll recesses into the grinding wheel with the **DRESSING FEED RATE Q1018** by the **DRESSING AMOUNT Q1013**.
4 If a **DWELL TIME IN REVS Q211** is defined, then it is executed by the control.
5 At **F PRE-POSITIONING Q253**, the control retracts the dressing roll to the **PRE-POSITION DIST. Q1025**.
6 Based on the **RECESSING OVERLAP Q510**, the control moves the dressing roll to the next recessing position in the Z axis of the dressing kinematic model.
7 The control repeats processes 3 to 6 until the entire grinding wheel is dressed.
8 At **F PRE-POSITIONING Q253**, the control retracts the dressing role to the **PRE-POSITION DIST. Q1025**.
9 The control moves to the starting position at rapid traverse.

> The control calculates the number or required recesses based on the width of the grinding wheel, the width of the dressing roll and the value of the parameter **RECESSING OVERLAP Q510**.

Notes

NOTICE

Danger of collision!

When you activate **FUNCTION DRESS BEGIN**, the control switches the kinematics. The grinding wheel becomes the workpiece. The axes may move in the opposite direction. There is a risk of collision during the execution of the function and during the subsequent machining!

- Activate the **FUNCTION DRESS** dressing mode only in mode **Program Run** mode or in **Single Block** mode
- Before starting **FUNCTION DRESS BEGIN**, position the grinding wheel near the dressing tool
- Once you have activated **FUNCTION DRESS BEGIN**, use exclusively cycles from HEIDENHAIN or from your machine tool builder
- In case the NC program is aborted or in case of a power interruption, check the traverse directions of the axes
- If necessary, program a kinematic switch-over

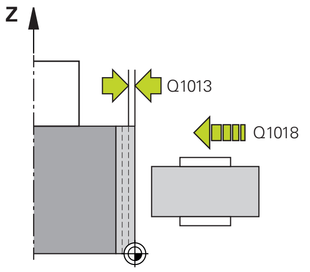
- Cycle **1018** is DEF-active.
- No coordinate transformations are allowed in dressing mode. The control displays an error message.
- The control does not graphically depict the dressing operation.
- If the width of the dressing roll is less than the width of the grinding wheel, then use the dressing strategy multiple recessing **Q1027=1**.
- If you program a **COUNTER FOR DRESSING Q1022**, then the control performs the dressing process only after reaching the defined counter from the tool management function. The control saves the **DRESS-N-D** and **DRESS-N-D-ACT** counters for every grinding wheel.

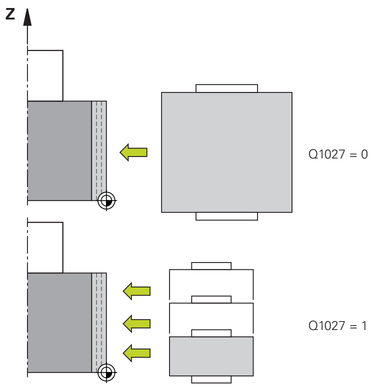
  Further information: "Dressing tool table tooldress.drs (option 156)", Page 1836
- At the end of every infeed run, the control corrects the tool data of the grinding tool and dressing tool.
- The control does not apply tool radius compensation in the dressing cycle.
- This cycle can only be run in dressing mode. The machine manufacturer may already have programmed the switch-over in the cycle sequence.

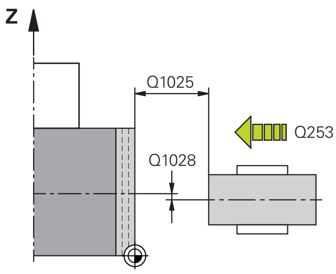
  Further information: "Simplified dressing with a macro", Page 228

Cycle parameters

Help graphic | **Parameter**







Q1013 Dressing amount?

Value used by the control for the dressing infeed.

Input: **0...9.9999**

Q1018 Feed rate for dressing?

Feed rate during the dressing procedure

Input: **0...99999**

Q1027 dressing strategy (0-1)?

Strategy during recessing with a dressing roll:

0: Recessing; the control executes a linear recessing movement. The grinding wheel width is less than the width of the dressing roll.

1: Multiple recessing; the control executes linear recessing movements. After infeeding to the dressing value, the control moves the dressing tool in the Z axis in the dressing kinematic model and infeeds again. The width of the grinding wheel is greater than the width of the dressing roll.

Input: **0**, **1**

Q1025 Distance for pre-positioning?

Distance between the grinding wheel and the dressing role during pre-positioning

Input: **0...9.9999**

Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min. while approaching the pre-position

Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Q211 Dwell time / 1/min?

Revolutions of the grinding wheel at the end of the recessing cut.

Input: **0...999.99**

Q1028 Offset of centers?

Offset of the middle of the dressing role relative to the middle of the grinding wheel. This offset takes effect in the Z axis of the dressing kinematic model. This value has an incremental effect.

If **Q1027=1**, then the control does not use a center offset.

Input: **-999.999...+999.999**

Help graphic	Parameter
	Q510 Overlap factor for recess width? With factor **Q510**, you influence the offset of the dressing roll in the Z axis of the dressing kinematic model. The control multiplies the factor with the value **CUTWIDTH** and offsets the dressing roll between the infeed runs by the calculated value. **1**: For every infeed run, the control recesses with the complete width of the dressing role. **Q510** takes effect only with **Q1027=1**. Input: **0.001...1**
	Q1026 Wear on dressing tool? Factor of the dressing value in order to define the wear on the dressing roll: **0**: The full dressing value is removed on the grinding wheel. **>0**: The factor is multiplied by the dressing value. The control takes the calculated value into account and assumes that this value will be lost during dressing due to wear on the dressing roll. The remaining dressing value is dressed on the grinding wheel. Input: **0...+0.99**
	Q1022 Dressing after number of calls? Number of cycle definitions after which the control performs the dressing process. Every cycle definition increments the counter **DRESS-N-D-ACT** of the grinding wheel in the tool manager. **0**: The control dresses the grinding wheel during every cycle definition in the NC program. **>0**: The control dresses the grinding wheel after this number of cycle definitions. Input: **0...99**
	Q330 Tool number or tool name? (optional) Number or name of the dressing tool. You can apply the tool directly from the tool table via selection in the action bar. **-1**: Dressing tool has been activated prior to the dressing cycle Input: **-1...99999.9**

Help graphic	Parameter
	Q1011 Factor for cutting speed? (optional, depends on the machine manufacturer) Factor by which the control changes the cutting speed for the dressing tool. The control handles the cutting speed of the grinding wheel. **0**: Parameter not programmed. **>0**: If the value is positive, then the dressing tool turns with the grinding wheel at the point of contact (opposite direction of rotation relative to grinding wheel). **<0**: If the value is negative, then the dressing tool turns against the grinding wheel (same direction of rotation of the grinding wheel). Input: **-99.999...99.999**

Example

```
11 CYCL DEF 1018 RECESSING WITH DRESSING ROLL ~
    Q1013=+1                ;DRESSING AMOUNT ~
    Q1018=+100              ;DRESSING FEED RATE ~
    Q1027=+0                ;DRESSING STRATEGY ~
    Q1025=+5                ;PRE-POSITION DIST. ~
    Q253=+1000              ;F PRE-POSITIONING ~
    Q211=+3                 ;DWELL TIME IN REVS ~
    Q1028=+1                ;CENTER OFFSET ~
    Q510=+0.8               ;RECESSING OVERLAP~
    Q1026=+0                ;WEAR FACTOR ~
    Q1022=+2                ;COUNTER FOR DRESSING ~
    Q330=-1                 ;TOOL ~
    Q1011=+0                ;FACTOR VC
```

14.5.12 Cycle 1021 CYLINDER, SLOW-STROKE GRINDING (option 156)

Application

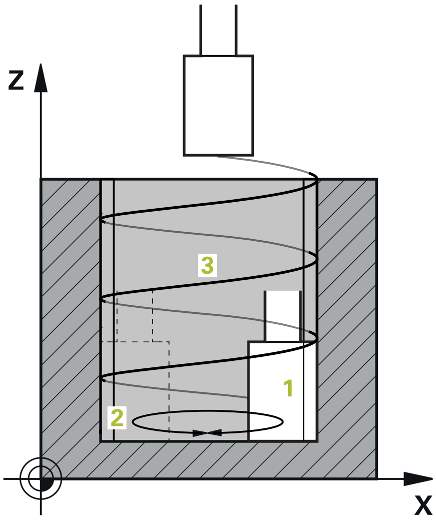
	Refer to your machine manual! This function must be enabled and adapted by the machine manufacturer.

With cycle **1021 CYLINDER, SLOW-STROKE GRINDING**, you can grind circular pockets or circular studs. The height of the cylinder can be considerably greater than the width of the grinding wheel. Through the reciprocating movement, the control can machine the entire height of the cylinder. The control executes multiple circular paths during the reciprocating stroke. In the processing, the reciprocating stroke and the circular paths overlap to form a helix. This process is equivalent to grinding with a slow stroke.

The lateral infeed cuts occur at the reversal points of the reciprocating stroke along the semi-circle. You can program the feed rate of the reciprocating stroke as the pitch of the helical path relative to the width of the grinding wheel.

You can also completely machine cylinders without overshoot, such as blind holes. This is done by programming idle runs at the reversal points of the reciprocating stroke.

Cycle sequence



1 The control positions the grinding tool above the cylinder, depending on **POCKET POSITION Q367**. The control now traverses the tool to the **CLEARANCE HEIGHT Q260** at rapid traverse.
2 The grinding tool traverses to the **SET-UP CLEARANCE Q200** at **F PRE-POSITIONING Q253**
3 The grinding tool traverses to the starting point in the tool axis. The starting point depends on the **MACHINING DIRECTION Q1031**, upper or lower reversal point of the reciprocating stroke.
4 The cycle starts the reciprocating stroke. At the **GRINDING FEED RATE Q207**, the control moves the grinding tool to the contour.

   Further information: "Feed rate for the reciprocating stroke", Page 872
5 The control delays the reciprocating stroke in the starting position.
6 Depending on **Q1021 ONE-SIDED INFEED**, the control infeeds the grinding tool in a semi-circle around the lateral infeed **Q534** **1**.
7 As needed, the control executes the defined idle runs **2** **Q211** or **Q210**.

   Further information: "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 872
8 The cycle continues the reciprocating movement. The grinding tool follows multiple circular paths. The reciprocating stroke overlays the circular paths in the direction of the tool axis to form a helix. You can influence the pitch of the helical path by the factor **Q1032**.
9 The circular paths **3** repeat themselves until the second reversal point of the reciprocating stroke is reached.
10 The control repeats steps 4 to 7 until the diameter of the finished part **Q223** or the oversize **Q14** is reached.
11 After the last lateral infeed run, the grinding wheel moves the number of programmed idle strokes **Q1020** if applicable.
12 The control stops the reciprocating stroke. The grinding tool leaves the cylinder on a semi-circular path to the safety clearance **Q200**.
13 The grinding tool moves to the **SET-UP CLEARANCE Q200** at the **F PRE-POSITIONING Q253** and then to the **CLEARANCE HEIGHT Q260** at rapid traverse.

> ℹ
> - In order for the grinding tool to completely machine the cylinder at the reversal points of the reciprocating stroke, you must define sufficient overshoot or idle runs.
> - The length of the reciprocating stroke arises from the **DEPTH Q201**, the **SURFACE OFFSET Q1030** and the wheel width **B**.
> - The starting point in the working plane is distant from the **FINISHED PART DIA. Q223** including **OVERSIZE AT START Q368** by the tool radius and the **SET-UP CLEARANCE Q200**.

Overshoot and idle runs to the reversal points of the reciprocating stroke

Path of the overshoot

Top	Bottom
This distance is defined in the parameter **Q1030 SURFACE OFFSET**.	You must add this distance to the depth and then define it in **Q201 DEPTH**.

If no overshoot is possible, such as with a pocket, program multiple idle runs at the reversal points of the reciprocating stroke (**Q210**, **Q211**). Select this number such that, after infeeding (half of a circular path), at least one circular path is traveled on the infed diameter. The number of idle runs is always based on a set feed-rate override of 100%.

> ℹ
> - HEIDENHAIN recommends moving with a feed-rate override of 100% or more. A feed-rate override of less than 100% no longer ensures that the cylinder will be completely machined at the reversal points.
> - For the definition of idle runs, HEIDENHAIN recommends defining at least a value of 1.5.

Feed rate for the reciprocating stroke

You can define the pitch per helical path (=360°) with the factor **Q1032**. Through this definition, the feed rate in mm or in inches/helical path (= 360°) can be derived for the reciprocating stroke.

The proportion of the **GRINDING FEED RATE Q207** to the feed rate of the reciprocating stroke plays a major role. If you deviate from a feed rate override of 100%, then ensure that the length of the reciprocating stroke during a circular path is less than the width of the grinding wheel.

> ℹ HEIDENHAIN recommends selecting a factor of at most 0.5.

Notes

	The overrides for the reciprocation movements can be changed by the machine manufacturer.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The last lateral infeed may be smaller depending on the input.
- The control does not depict the reciprocating movement in the simulation. The reciprocating movement is depicted in the simulation graphics in the **Program run, single block** and **Program run, full sequence** operating modes.
- You can also execute this cycle with a milling cutter. In the case of a milling cutter, the tooth length **LCUTS** equals the width of the grinding wheel.
- Please note that the cycle takes **M109** into account. The **GRINDING FEED RATE Q207** in the status display during program run in the case of a pocket is therefore smaller than in the case of a stud. The control shows the feed rate of the center point path of the grinding tool, including the reciprocating stroke.

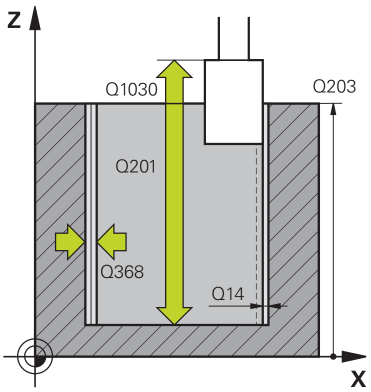
  Further information: "Adapting the feed rate for circular paths with M109", Page 1236

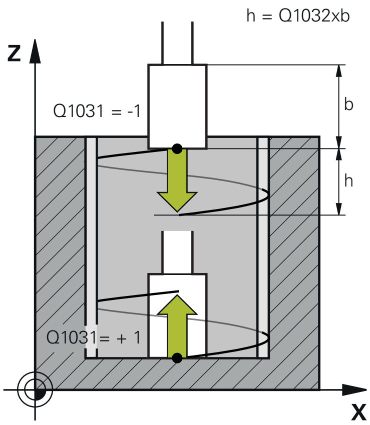
Notes on programming

- The control assumes that the bottom of the cylinder has a floor. For this reason, you can define an overshoot in **Q1030** only at the surface. If you machine a through hole, for example, then you must take into account the lower overshoot in **DEPTH Q201**.

  Further information: "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 872
- If the grinding wheel is wider than **DEPTH Q201** and the **SURFACE OFFSET Q1030**, then the control issues a **No swing stroke** error message. In this case, the resulting reciprocating stroke would be equal to 0.

Cycle parameters

Help graphic	Parameter
	Q650 Type of figure? Geometry of the figure: **0**: Pocket **1**: Island Input: **0**, **1**
	Q223 Finished part diameter? Diameter of the fully machined cylinder Input: **0...99999.9999**
 	**Q368 Side oversize before machining?** Lateral oversize that is present prior to the grinding operation. This value must be greater than **Q14**. This value has an incremental effect. Input: **-0.9999...+99.9999**
	Q14 Finishing allowance for side? Lateral oversize that is to remain after machining. This allowance must be less than **Q368**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q367 Position of pocket (0/1/2/3/4)? Position of the figure relative to the position of the tool during the cycle call: **0**: Tool pos. = Center of figure **1**: Tool pos. = Quadrant transition at 90° **2**: Tool pos. = Quadrant transition at 0° **3**: Tool pos. = Quadrant transition at 270° **4**: Tool pos. = Quadrant transition at 180° Input: **0**, **1**, **2**, **3**, **4**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q1030 Offset to surface? Position of the upper edge of the tool on the surface. The offset serves as the overshoot path on the surface for the reciprocating stroke. The value has an absolute effect. Input: **0...999.999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**

Help graphic	Parameter
	**Q1031 Machining direction?** Definition of the start position. The direction of the first reciprocating stroke arises from this. **–1** or **0**: The starting position is on the surface. The reciprocating stroke begins in the negative direction. **+1**: The starting position is at the cylinder floor. The reciprocating stroke begins in the positive direction. Input: **-1**, **0**, **+1**
	Q1021 One-sided infeed (0/1)? Position at which the lateral infeed occurs: **0**: Lower and upper lateral infeed **1**: One-sided infeed depending on **Q1031** ■ If **Q1031 = -1**, then the lateral infeed is performed above. ■ If **Q1031 = +1**, then the lateral infeed is performed below. Input: **0**, **1**
	Q534 Lateral infeed? Amount by which the grinding tool is laterally infed. Input: **0.0001...99.9999**
	Q1020 Number of idle strokes? Number of idle strokes after the last lateral infeed without material removal. Input: **0...99**
	Q1032 Factor for pitch of helix? The pitch per helical path (= 360°) arises from the factor **Q1032**. **Q1032** is multiplied by the width **B** of the grinding tool. The feed rate for the reciprocating stroke is influenced by the pitch of the helical path. **Further information:** "Feed rate for the reciprocating stroke", Page 872 Input: **0.000...1000**
	Q207 Feed rate for grinding? Traversing speed of the tool during grinding of the contour in mm/min Input: **0...99999.999** or **FAUTO**, **FU**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the **DEPTH Q201**. The feed rate has an effect below the **SURFACE COORDINATE Q203**. Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q15 Up-cut / climb grinding (-1/+1)? Define the type of contour grinding: **+1**: Climb grinding **-1** or **0**: Up-cut grinding Input: **-1**, **0**, **+1**
	Q260 Clearance height? Absolute height at which no collision can occur with the workpiece. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q211 Idle runs at depth? Number of idle runs at the lower reversal point of the reciprocating stroke. **Further information:** "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 872. Input: **0...99.99**
	Q210 Idle runs at top? Number of idle runs at the upper reversal point of the reciprocating stroke. **Further information:** "Overshoot and idle runs to the reversal points of the reciprocating stroke", Page 872. Input: **0...99.99**

Example

```
11 CYCL DEF 1021 CYLINDER, SLOW-STROKE GRINDING ~
    Q650=+0                    ;FIGURE TYPE ~
    Q223=+50                   ;FINISHED PART DIA. ~
    Q368=+0.1                  ;OVERSIZE AT START ~
    Q14=+0                     ;ALLOWANCE FOR SIDE ~
    Q367=+0                    ;POCKET POSITION ~
    Q203=+0                    ;SURFACE COORDINATE ~
    Q1030=+2                   ;VERSATZ OBERFLAECHE ~
    Q201=-20                   ;DEPTH ~
    Q1031=+1                   ;MACHINING DIRECTION ~
    Q1021=+0                   ;ONE-SIDED INFEED ~
    Q534=+0.01                 ;LATERAL INFEED ~
    Q1020=+0                   ;IDLE STROKES ~
    Q1032=+0.5                 ;FAKTOR ZUSTELLUNG ~
    Q207=+2000                 ;GRINDING FEED RATE ~
    Q253=+750                  ;F PRE-POSITIONING ~
    Q15=-1                     ;TYPE OF GRINDING ~
    Q260=+100                  ;CLEARANCE HEIGHT ~
    Q200=+2                    ;SET-UP CLEARANCE ~
    Q211=+0                    ;IDLE RUNS AT DEPTH ~
    Q210=+0                    ;IDLE RUNS AT TOP
```

14.5.13 Cycle 1022 CYLINDER, FAST-STROKE GRINDING (option 156)

Application

> Refer to your machine manual!
> This function must be enabled and adapted by the machine manufacturer.

With the cycle **1022 CYLINDER, FAST STROKE GRINDING**, you can grind circular pockets and circular studs. In the process, the control executes circular and helical paths in order to completely machine the cylinder surface. In order to achieve the required accuracy and surface quality, you can overlay the movement with a reciprocating stroke. The feed rate of the reciprocating stroke is usually so large that multiple reciprocating strokes per circular path are executed. This is equivalent to grinding with a rapid stroke. The lateral infeeds occur above or below depending on the definition. You can program the feed rate of the reciprocating stroke in the cycle.

Cycle sequence

1 The control positions the tool above the cylinder based on the **POCKET POSITION Q367**. At **FMAX**, the control then moves the tool to the **CLEARANCE HEIGHT Q260**.
2 At **FMAX**, the tool moves to the starting point in the working plane and then at **F PRE-POSITIONING Q253** to the **SET-UP CLEARANCE Q200**.
3 The grinding tool moves to the starting point in the tool axis. The starting point depends on the **MACHINING DIRECTION Q1031**. If you have defined a reciprocating stroke in **Q1000**, then the control starts the reciprocating stroke.
4 Depending on the parameter **Q1021**, the control laterally infeeds the grinding tool. The control then infeeds in the tool axis.

   Further information: "Infeed", Page 878
5 If the final depth has been reached, then the grinding tool moves for another full circle without a tool axis infeed.
6 The control repeats steps 4 and 5 until the diameter of the finished part **Q223** or the oversize **Q14** has been reached.
7 After the last infeed run, the grinding tool executes the **IDLE RUNS, CONT. END Q457**.
8 The grinding tool leaves the cylinder on a semi-circular path to the safety clearance **Q200** and stops the reciprocating stroke.
9 At **F PRE-POSITIONING Q253**, the control moves the tool to the **SAFETY CLEARANCE Q200** and then in rapid traverse to the **CLEARANCE HEIGHT Q260**.

Infeed

1 The control infeeds the grinding tool in a semi-circle to the **LATERAL INFEED Q534**.
2 The grinding tool executes a full circle and performs any programmed **IDLE RUNS, CONTOUR Q456**.
3 If the area to be traversed in the tool axis is greater than the grinding wheel width **B**, then the cycle moves in a helical path.

Helical path

You can influence the helical path via a pitch in the parameter **Q1032**. The pitch per helical path (= 360°) is relative to the grinding wheel width.

The number of helical paths (= 360°) depends on the pitch and the **DEPTH Q201**. The smaller the pitch, the more helical paths (= 360°) there are.

Example:

- Grinding wheel width **B** = 20 mm
- **Q201 DEPTH** = 50 mm
- **Q1032 PITCH FACTOR** (pitch) = 0.5

The control calculates the relationship between the pitch relative to the grinding wheel width.

Pitch per helical path = $20mm * 0.5 = 10mm$

The control covers the distance of 10 mm in the tool axis within a helix. The **DEPTH Q201** and the pitch per helical path result in five helical paths.

Number of helical paths = $\frac{50mm}{10mm} = 5$

Notes

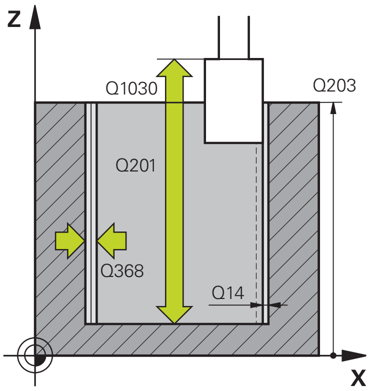
	The overrides for the reciprocation movements can be changed by the machine manufacturer.

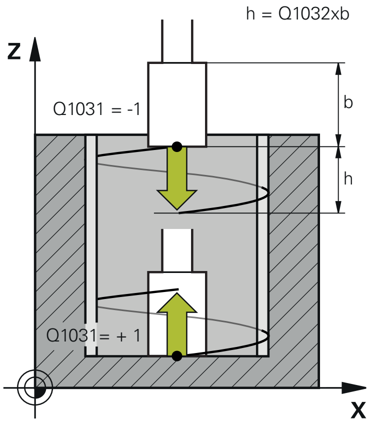
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control always starts the reciprocating stroke in the positive direction.
- The last lateral infeed may be smaller depending on the input.
- The control does not depict the reciprocating movement in the simulation. The reciprocating movement is depicted in the simulation graphics in the **Program run, single block** and **Program run, full sequence** operating modes.
- You can also execute this cycle with a milling cutter. In the case of a milling cutter, the tooth length **LCUTS** equals the width of the grinding wheel.

Notes on programming

- The control assumes that the bottom of the cylinder has a floor. For this reason, you can define an overshoot in **Q1030** only at the surface. If you machine a through hole, for example, then you must take into account the lower overshoot in **DEPTH Q201**.
- If **Q1000=0**, then the control does not execute a superimposed reciprocating movement.

Cycle parameters

Help graphic	Parameter
	Q650 Type of figure? Geometry of the figure: **0**: Pocket **1**: Island Input: **0**, **1**
	Q223 Finished part diameter? Diameter of the fully machined cylinder Input: **0...99999.9999**
 	**Q368 Side oversize before machining?** Lateral oversize that is present prior to the grinding operation. This value must be greater than **Q14**. This value has an incremental effect. Input: **-0.9999...+99.9999**
	Q14 Finishing allowance for side? Lateral oversize that is to remain after machining. This allowance must be less than **Q368**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q367 Position of pocket (0/1/2/3/4)? Position of the figure relative to the position of the tool during the cycle call: **0**: Tool pos. = Center of figure **1**: Tool pos. = Quadrant transition at 90° **2**: Tool pos. = Quadrant transition at 0° **3**: Tool pos. = Quadrant transition at 270° **4**: Tool pos. = Quadrant transition at 180° Input: **0**, **1**, **2**, **3**, **4**
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q1030 Offset to surface? Position of the upper edge of the tool on the surface. The offset serves as the overshoot path on the surface for the reciprocating stroke. The value has an absolute effect. Input: **0...999.999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**

Help graphic	Parameter
	**Q1031 Machining direction?** Definition of the machining direction. The starting position arises from this. **-1** or **0**: The control machines the contour from up to down during the first infeed cut. **+1**: The control machines the contour from up to down during the first infeed cut. Input: **-1**, **0**, **+1**
	Q534 Lateral infeed? Amount by which the grinding tool is laterally infed. Input: **0.0001...99.9999**
	Q1032 Factor for pitch of helix? You can define the pitch of the helical path (= 360°) with the factor **Q1032**. This results in the infeed depth per helical path (= 360°). **Q1032** is multiplied by the width **B** of the grinding tool. Input: **0.000...1000**
	Q456 Idle runs around contour? Number of times the grinding tool executes the contour without removing material after every infeed. Input: **0...99**
	Q457 Idle runs at contour end? Number of times the grinding tool executes the contour without material removal after the last infeed. Input: **0...99**
	Q1000 Length of reciprocating stroke? Length of the reciprocating movement, parallel to the active tool axis **0**: The control does not perform a reciprocating motion. Input: **0...9999.9999**
	Q1001 Feed rate for reciprocation? Speed of the reciprocating stroke in mm/min Input: **0...999999**
	Q1021 One-sided infeed (0/1)? Position at which the lateral infeed occurs: **0**: Lower and upper lateral infeed **1**: One-sided infeed depending on **Q1031** ▪ If **Q1031 = -1**, then the lateral infeed is performed above. ▪ If **Q1031 = +1**, then the lateral infeed is performed below. Input: **0**, **1**

Help graphic	Parameter
	Q207 Feed rate for grinding? Traversing speed of the tool during grinding of the contour in mm/min Input: **0...99999.999** or **FAUTO**, **FU**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the **DEPTH Q201**. The feed rate has an effect below the **SURFACE COORDINATE Q203**. Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q15 Up-cut / climb grinding (-1/+1)? Define the type of contour grinding: **+1**: Climb grinding **-1** or **0**: Up-cut grinding Input: **-1**, **0**, **+1**
	Q260 Clearance height? Absolute height at which no collision can occur with the workpiece. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 CYCL DEF 1022 CYLINDER, FAST-STROKE GRINDING ~
    Q650=+0                 ;FIGURE TYPE ~
    Q223=+50                ;FINISHED PART DIA. ~
    Q368=+0.1               ;OVERSIZE AT START ~
    Q14=+0                  ;ALLOWANCE FOR SIDE ~
    Q367=+0                 ;POCKET POSITION ~
    Q203=+0                 ;SURFACE COORDINATE ~
    Q1030=+2                ;SURFACE OFFSET ~
    Q201=-20                ;DEPTH ~
    Q1031=-1                ;MACHINING DIRECTION ~
    Q534=+0.05              ;LATERAL INFEED ~
    Q1032=+0.5              ;PITCH FACTOR ~
    Q456=+0                 ;IDLE RUNS, CONTOUR ~
    Q457=+0                 ;IDLE RUNS, CONT. END ~
    Q1000=+5                ;RECIPROCATING STROKE ~
    Q1001=+5000             ;RECIP. FEED RATE ~
    Q207=+50                ;GRINDING FEED RATE ~
    Q253=+750               ;F PRE-POSITIONING ~
    Q15=+1                  ;TYPE OF GRINDING ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q200=+2                 ;SET-UP CLEARANCE
```

14.5.14 Cycle 1025 GRINDING CONTOUR (option 156)

Application

Use Cycle **1025 GRINDING CONTOUR** in combination with Cycle **14 CONTOUR** to grind open and closed contours.

Cycle run

1 The control first moves the tool at rapid traverse to the starting position in the X and Y directions and then to clearance height **Q260**.
2 The tool uses rapid traverse to move to set-up clearance **Q200** above the coordinate surface.
3 From there, it moves at the pre-positioning feed rate **Q253** to the depth **Q201**.
4 If programmed, the control performs the approach movement.
5 The cycle starts with the first stepover **Q534**.
6 If programmed, the control performs the number of idle runs **Q456** after each infeed.
7 This process (steps 5 and 6) is repeated until the contour or finishing allowance **Q14** has been reached.
8 After the last infeed, the specified number of air strokes at contour end **Q457** are performed.
9 The control performs the optional departure movement.
10 Finally, the tool is moved at rapid traverse to the clearance height.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The last stepover may be smaller depending on the input.

Note on programming

- If you want to program a reciprocating stroke, you need to define and start it before executing this cycle.

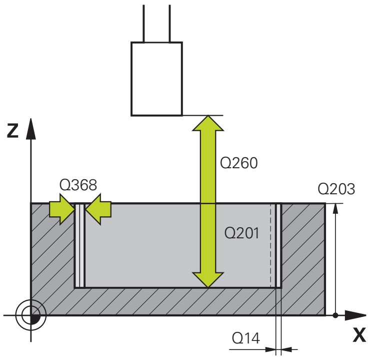
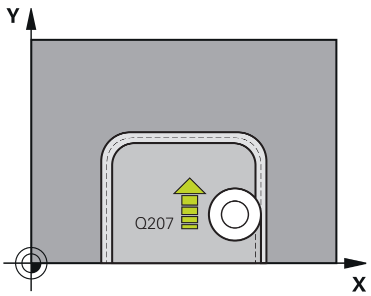
Open contour

- Approach and departure movements for the contour can be programmed using **APPR** and **DEP** or Cycle **270**.

Closed contour

- In the case of a closed contour, only Cycle **270** is available for programming approach and departure movements.
- When grinding a closed contour, it is not possible to alternate between climb and up-cut grinding (**Q15 = 0**). The control issues an error message.
- If you programmed approach and departure movements, the starting position will shift with every infeed. If no approach and departure movements have been programmed, the control automatically generates a vertical movement and the starting position on the contour will not shift.

Cycle parameters

Help graphic	Parameter
	**Q203 Workpiece surface coordinate?** Coordinate on the workpiece surface referenced to the active datum. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q201 Depth? Distance between the workpiece surface and the contour floor. This value has an incremental effect. Input: **-99999.9999...+0**
	Q14 Finishing allowance for side? Lateral oversize that is to remain after machining. This allowance must be less than **Q368**. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q368 Side oversize before machining? Lateral oversize that is present prior to the grinding operation. This value must be greater than **Q14**. This value has an incremental effect. Input: **-0.9999...+99.9999**
	Q534 Lateral infeed? Amount by which the grinding tool is laterally infed. Input: **0.0001...99.9999**
	Q456 Idle runs around contour? Number of times the grinding tool executes the contour without removing material after every infeed. Input: **0...99**
	**Q457 Idle runs at contour end?** Number of times the grinding tool executes the contour without material removal after the last infeed. Input: **0...99**
	Q207 Feed rate for grinding? Traversing speed of the tool during grinding of the contour in mm/min Input: **0...99999.999** or **FAUTO**, **FU**
	Q253 Feed rate for pre-positioning? Traversing speed of the tool when approaching the **DEPTH Q201**. The feed rate has an effect below the **SURFACE COORDINATE Q203**. Input in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q15 Up-cut / climb grinding (-1/+1)? Define the machining direction of the contours: **+1**: Climb grinding **-1**: Up-cut grinding **0**: Alternating between climb grinding and up-cut grinding Input: **-1**, **0**, **+1**
	Q260 Clearance height? Absolute height at which no collision can occur with the workpiece. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

```
11 CYCL DEF 1025 GRINDING CONTOUR ~
   Q203=+0              ;SURFACE COORDINATE ~
   Q201=-20             ;DEPTH ~
   Q14=+0               ;ALLOWANCE FOR SIDE ~
   Q368=+0.1            ;OVERSIZE AT START ~
   Q534=+0.05           ;LATERAL INFEED ~
   Q456=+0              ;IDLE RUNS, CONTOUR ~
   Q457=+0              ;IDLE RUNS, CONT. END ~
   Q207=+200            ;GRINDING FEED RATE ~
   Q253=+750            ;F PRE-POSITIONING ~
   Q15=+1               ;TYPE OF GRINDING ~
   Q260=+100            ;CLEARANCE HEIGHT ~
   Q200=+2              ;SET-UP CLEARANCE
```

14.5.15 Cycle 1030 ACTIVATE WHEEL EDGE (option 156)

Application

Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1030 ACTIVATE WHEEL EDGE** to activate the desired wheel edge. This means that you can change or update the reference point or reference edge. When dressing, you set the workpiece datum to the corresponding wheel edge with this cycle.

For this cycle, a distinction is made between grinding **(FUNCTION MODE MILL / TURN)** and dressing **(FUNCTION DRESS BEGIN / END)**.

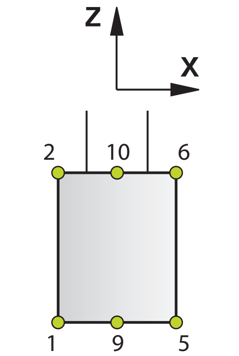
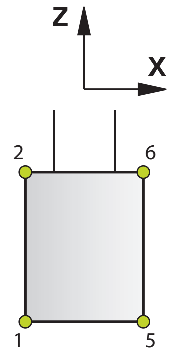
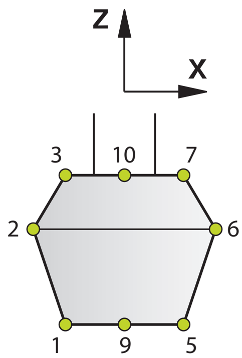
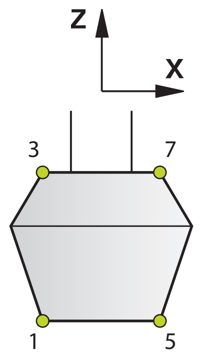
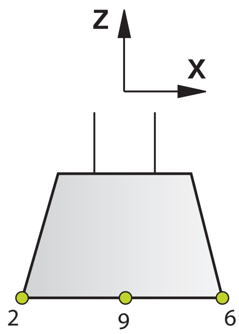
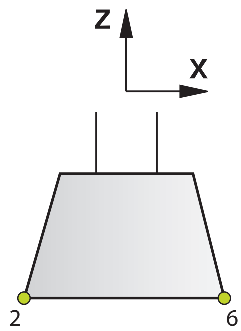
Notes

- This cycle is only permitted in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes if a grinding tool has been activated.
- Cycle **1030** is DEF-active.

Cycle parameters

Help graphic	Parameter
	Q1006 Edge of grinding wheel? Definition of the edge of the grinding tool

Selection of the grinding wheel edges

	Grinding	Dressing
Grinding pin		
Special grinding pin		
Cup wheel		

Example

```
11 CYCL DEF 1030 ACTIVATE WHEEL EDGE ~
    Q1006=+9                    ;WHEEL EDGE
```

14.5.16 Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)

Application

Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1032 GRINDING WHL LENGTH COMPENSATION** to define the overall length of a grinding tool. This cycle will modify compensation or basic data, depending on whether an initial dressing operation (**INIT_D**) was carried out or not. This cycle will insert the values automatically at the correct locations in the tool table.

If initial dressing has not been performed (INIT_D_OK = 0), then you can change the basic data. Basic data affect both grinding and dressing.

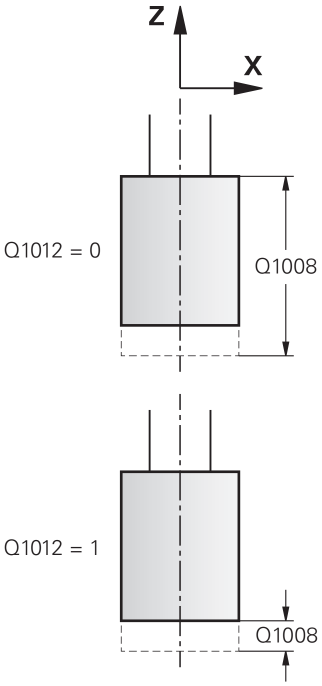
If initial dressing has already been carried out (checkbox for **INIT_D** is enabled), you can edit the compensation data. Compensation data affect grinding only.

Further information: "Dressing", Page 227

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Cycle **1032** is DEF-active.

Cycle parameters

Help graphic	Parameter
	**Q1012 Compens. values (0=abs./1=inc.)?** Definition of the entered length dimension **0**: Entry of the absolute length **1**: Entry of the incremental length Input: **0**, **1**
	Q1008 Comp. value outside edge length? Amount by which the tool is corrected lengthwise based on **Q1012** or by which the tool data are entered without correction. If **Q1012** equals **0**, then the absolute length must be entered. If **Q1012** equals **1**, then the incremental length must be entered. Input: **-999.999...+999.999**
	Q330 Tool number or tool name? Number of name of the grinding tool. Via a selection in the action bar, you have the option of applying the tool directly from the tool table. **-1**: The active tool from the tool spindle is used. Input: **-1...99999.9**

Example

```
11 CYCL DEF 1032 GRINDING WHL LENGTH COMPENSATION ~
    Q1012=+1                ;INCR. COMPENSATION ~
    Q1008=+0                ;COMP. OUTSIDE LENGTH ~
    Q330=-1                 ;TOOL
```

14.5.17 Cycle 1033 GRINDING WHL RADIUS COMPENSATION (option 156)

Application

Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

Use Cycle **1033 GRINDING WHL RADIUS COMPENSATION** to define the radius of a grinding tool. This cycle will modify compensation or basic data, depending on whether an initial dressing operation (**INIT_D**) was carried out or not. This cycle will insert the values automatically at the correct locations in the tool table.

If initial dressing has not been performed (INIT_D_OK = 0), then you can change the basic data. Basic data affect both grinding and dressing.

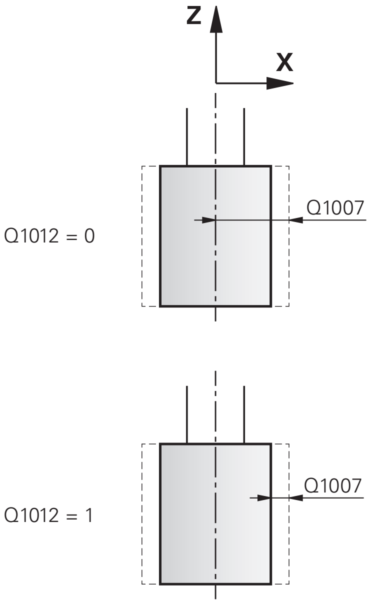
If initial dressing has already been carried out (checkbox for **INIT_D** is enabled), you can edit the compensation data. Compensation data affect grinding only.

Further information: "Dressing", Page 227

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Cycle **1033** is DEF-active.

Cycle parameters

Help graphic	Parameter
	**Q1012 Compens. values (0=abs./1=inc.)?** Definition of the entered radius dimension **0**: Entry of the absolute radius **1**: Entry of the incremental radius Input: **0**, **1**
	Q1007 Compensation value for radius? Dimension by which the tool radius is compensated for based on **Q1012**. If **Q1012** equals **0**, then the absolute radius must be entered. If **Q1012** equals **1**, then the incremental radius must be entered. Input: **-999.9999...+999.9999**
	Q330 Tool number or tool name? Number of name of the grinding tool. Via a selection in the action bar, you have the option of applying the tool directly from the tool table. **-1**: The active tool from the tool spindle is used. Input: **-1...99999.9**

Example

```
11 CYCL DEF 1033 GRINDING WHL RADIUS COMPENSATION ~
    Q1012=+1                ;INCR. COMPENSATION ~
    Q1007=+0                ;RADIUS COMPENSATION ~
    Q330=-1                 ;TOOL
```

14.5.18 Programming examples

Example of grinding cycles

This programming example illustrates how to machine with a grinding tool.

The NC program uses the following grinding cycles:

- Cycle **1000 DEFINE RECIP. STROKE**
- Cycle **1002 STOP RECIP. STROKE**
- Cycle **1025 GRINDING CONTOUR**

Program sequence

- Start milling mode
- Tool call: Grinding pin
- Define Cycle **1000 DEFINE RECIP. STROKE**
- Define Cycle **14 CONTOUR**
- Define Cycle **1025 GRINDING CONTOUR**
- Define Cycle **1002 STOP RECIP. STROKE**

0 BEGIN PGM GRINDING_CYCLE MM	
1 BLK FORM 0.1 Z X-9.6 Y-25.1 Z-33	
2 BLK FORM 0.2 X+9.6 Y+25.1 Z+1	
3 FUNCTION MODE MILL	
4 TOOL CALL 501 Z S20000	; tool call: grinding tool
5 L Z+30 R0 F1000 M3	
6 CYCL DEF 1000 DEFINE RECIP. STROKE ~	
Q1000=+13 ;RECIPROCATING STROKE ~	
Q1001=+25000 ;RECIP. FEED RATE ~	
Q1002=+1 ;RECIPROCATION TYPE ~	
Q1004=+1 ;START RECIP. STROKE	
7 CYCL DEF 14.0 CONTOUR	
8 CYCL DEF 14.1 CONTOUR LABEL1 /2	
9 CYCL DEF 1025 GRINDING CONTOUR ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q201=-12 ;DEPTH ~	
Q14=+0 ;ALLOWANCE FOR SIDE ~	
Q368=+0.2 ;OVERSIZE AT START ~	
Q534=+0.05 ;LATERAL INFEED ~	
Q456=+2 ;IDLE RUNS, CONTOUR ~	
Q457=+3 ;IDLE RUNS, CONT. END ~	
Q207=+200 ;GRINDING FEED RATE ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q15=+1 ;TYPE OF GRINDING ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q200=+2 ;SET-UP CLEARANCE	
10 CYCL CALL	; cycle call: grinding contour
11 L Z+50 R0 FMAX	

12 CYCL DEF 1002 STOP RECIP. STROKE ~	
Q1005=+1 ;CLEAR RECIP. STROKE ~	
Q1010=+0 ;RECIP.STROKE STOPPOS	
13 L Z+250 R0 FMAX	
14 L C+0 R0 FMAX M92	
15 M30	; end of program
16 LBL 1	; contour subprogram 1
17 L X+3 Y-23 RL	
18 L X-3	
19 CT X-9 Y-16	
20 CT X-7 Y-10	
21 CT X-7 Y+10	
22 CT X-9 Y+16	
23 CT X-3 Y+23	
24 L X+3	
25 CT X+9 Y+16	
26 CT X+7 Y+10	
27 CT X+7 Y-10	
28 CT X+9 Y-16	
29 CT X+3 Y-23	
30 LBL 0	
31 LBL 2	; contour subprogram 2
32 L X-25 Y-40 RR	
33 L Y+40	
34 L X+25	
35 L Y-40	
36 L X-25	
37 LBL 0	
38 END PGM GRINDING_CYCLE MM	

Example of dressing cycles

This programming example illustrates dressing mode.

The NC program uses the following grinding cycles:

- Cycle **1030 ACTIVATE WHEEL EDGE**
- Cycle **1010 DRESSING DIAMETER**

Program sequence

- Start milling mode
- Tool call: Grinding pin
- Define Cycle **1030 ACTIVATE WHEEL EDGE**
- Tool call: Dressing tool (no mechanical tool change; only a calculated switch-over)
- Cycle **1010 DRESSING DIAMETER**
- Activate **FUNCTION DRESS END**

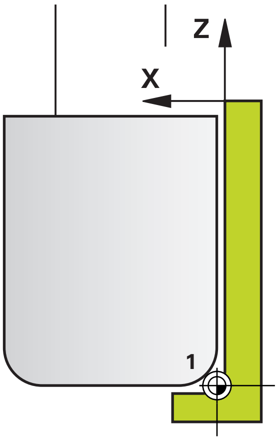
0 BEGIN PGM DRESS_CYCLE MM	
1 BLK FORM 0.1 Z X-9.6 Y-25.1 Z-33	
2 BLK FORM 0.2 X+9.6 Y+25.1 Z+1	
3 FUNCTION MODE MILL	
4 TOOL CALL 501 Z S20000	; tool call, grinding wheel
5 M140 MB MAX	
6 L Z+200 R0 FMAX M3	
7 FUNCTION DRESS BEGIN	; activate dressing procedure
8 CYCL DEF 1030 ACTIVATE WHEEL EDGE ~	
Q1006=+5 ;WHEEL EDGE	
9 TOOL CALL 507	; tool call, dressing tool
10 L X+5 R0 F2000	
11 L Y+0 R0	
12 L Z-5 M8	
13 CYCL DEF 1010 DRESSING DIAMETER ~	
Q1013=+0 ;DRESSING AMOUNT ~	
Q1018=+300 ;DRESSING FEED RATE ~	
Q1016=+1 ;DRESSING STRATEGY ~	
Q1019=+2 ;NUMBER INFEEDS ~	
Q1020=+3 ;IDLE STROKES ~	
Q1022=+0 ;COUNTER FOR DRESSING ~	
Q330=-1 ;TOOL ~	
Q1011=+0 ;FACTOR VC	
14 FUNCTION DRESS END	; deactivate dressing procedure
15 M30	; end of program
16 END PGM DRESS_CYCLE MM	

Example of a profile program

Grinding wheel edge no. 1

This example program is for dressing a profile of a grinding wheel. The grinding wheel is curved by the amount of a radius on its outer side.

The contour must be closed. The active edge is defined as the datum of the profile. You program the traverse path. (This is the green area in the illustration.)



Data to be used:

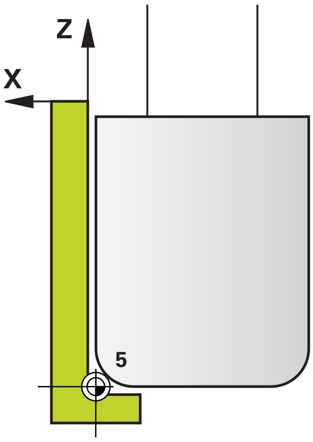
- Grinding wheel edge: 1
- Retraction amount: 5 mm
- Width of the pin: 40 mm
- Corner radius: 2 mm
- Depth: 6 mm

0 BEGIN PGM 11 MM	
1 L X-5 Z-5 R0 FMAX	; approach starting position
2 L Z+45 RL FMAX	; approach starting position
3 L X+0 FQ1018	; Q1018 = Dressing feed rate
4 L Z+0 FQ1018	; approach radius edge
5 RND R2 FQ1018	; rounding
6 L X+6 FQ1018	; approach final position X
7 L Z-5 FQ1018	; approach final position Z
8 L X-5 Z-5 R0 FMAX	; approach starting position
9 END PGM 11 MM	

Grinding wheel edge no. 5

This example program is for dressing a profile of a grinding wheel. The grinding wheel is curved by the amount of a radius on its outer side.

The contour must be closed. The active edge is defined as the datum of the profile. You program the traverse path. (This is the green area in the illustration.)



Data to be used:

- Grinding wheel edge: 5
- Retraction amount: 5 mm
- Width of the pin: 40 mm
- Corner radius: 2 mm
- Depth: 6 mm

0 BEGIN PGM 12 MM	
1 L X+5 Z-5 R0 FMAX	; approach starting position
2 L Z+45 RR FMAX	; approach starting position
3 L X+0 FQ1018	; Q1018 = Dressing feed rate
4 L Z+0 FQ1018	; approach radius edge
5 RND R2 FQ1018	; rounding
6 L X-6 FQ1018	; approach final position X
7 L Z-5 FQ1018	; approach final position Z
8 L X+5 Z-5 R0 FMAX	; approach starting position
9 END PGM 11 MM	

14.6 Cycles for gear cutting

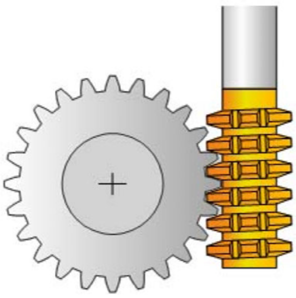
14.6.1 Overview

Cycle			Further information
880	**GEAR HOBBING** (option 50 & 131) ■ Description of the geometry and the tool ■ Selection of machining strategy and machining side	**CALL-**active	"Cycle 880 GEAR HOBBING (option 131)"
285	**DEFINE GEAR** (option 157) ■ Define the geometry of the gear wheel	**DEF-**active	"Cycle 285 DEFINE GEAR (option 157)"
286	**GEAR HOBBING** (option 157) ■ Definition of the tool data ■ Selection of the machining strategy and side ■ Possibility of using the entire cutting edge	**CALL-**active	"Cycle 286 GEAR HOBBING (option 157)"
287	**GEAR SKIVING** (option 157) ■ Definition of the tool data ■ Selection of the machining side ■ Definition of the first and last infeed ■ Definition of the number of cuts	**CALL-**active	"Cycle 287 GEAR SKIVING option 157"

14.6.2 Cycle 880 GEAR HOBBING (option 131)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



With Cycle **880 GEAR HOBBING**, you can machine external cylindrical gears or helical gears with any angles. In the cycle you first define the **gear** and then the **tool** with which the gear is to be machined. You can select the machining strategy and the machining side in the cycle. The machining process for gear hobbing is performed with a synchronized rotary motion of the tool spindle and rotary table. In addition, the gear hob moves along the workpiece in axial direction.

While Cycle **880 GEAR HOBBING** is active, the coordinate system might be rotated. It is therefore essential to program Cycle **801 RESET ROTARY COORDINATE SYSTEM** and **M145** after the end of the cycle.

Cycle sequence

1 The control positions the tool in the tool axis to clearance height **Q260** at the feed rate FMAX. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.
2 Before tilting the working plane, the control positions the tool in X to a safe coordinate at the FMAX feed rate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control then tilts the working plane at the feed rate **Q253**; **M144** is internally active in the cycle
4 The control positions the tool at the feed rate FMAX to the starting point in the working plane.
5 The control then moves the tool in the tool axis at the feed rate **Q253** to set-up clearance **Q460**.
6 The control now moves the tool at the defined feed rate **Q478** (for roughing) or **Q505** (for finishing) to hob the workpiece in longitudinal direction. The area to be machined is limited by the starting point in Z **Q551**+**Q460** and the end point in Z **Q552**+**Q460**.
7 When the control reaches the end point, it retracts the tool at the feed rate **Q253** and positions it back to the starting point
8 The control repeats the steps 5 to 7 until the defined gear is completed.
9 Finally the control positions the tool to the clearance height **Q260** at the feed rate FMAX
10 The machining operation ends in the tilted system.
11 Now you need to move the tool to a safe height and reset the tilting of the working plane.
12 It is essential that you now program Cycle **801 RESET ROTARY COORDINATE SYSTEM** and **M145**

Notes

NOTICE

Danger of collision!

If you do not position the tool to a safe position, a collision may occur between the tool and workpiece (fixtures) during tilting.

- Pre-position the tool so that it is already on the desired machining side **Q550**.
- Move the tool to a safe position on this machining side

NOTICE

Danger of collision!

If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining. The starting point in Z and the end point in Z are extended by the set-up clearance **Q460**!

- Clamp the workpiece out of the fixtures far enough to prevent a danger of collision between the tool and the fixtures
- Clamp the workpiece in such a way that its protrusion from the fixture will not cause any collision when the tool is automatically moved to the starting or end point using a path that is extended by the set-up clearance **Q460**

NOTICE

Danger of collision!

Depending on whether you use **M136** or not, the feed rate values will be interpreted differently by the control. If the programmed feed rate was too high, the workpiece might be damaged.

- If you program **M136** explicitly before the cycle, the control will interpret the feed rates in the cycle in mm/rev.
- If you do not program **M136** before the cycle, the control will interpret the feed rates in the cycle in mm/min.

NOTICE

Danger of collision!

If you do not reset the coordinate system after Cycle **880**, the precession angle set by the cycle will remain in effect.

- Make sure to program Cycle **801** after Cycle **880** in order to reset the coordinate system.
- Make sure to program Cycle **801** after a program abort in order to reset the coordinate system.

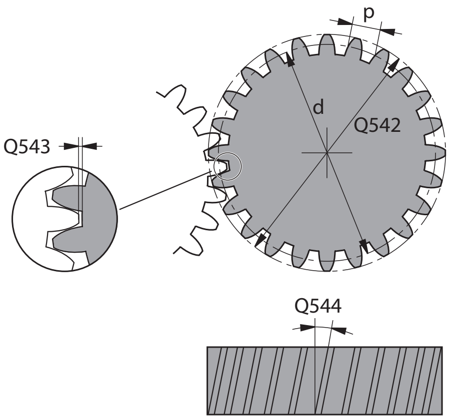
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The cycle is CALL-active.
- Define the tool as a milling cutter in the tool table.
- Before programming the cycle call, set the datum to the center of rotation.

In order to avoid that the maximum permissible spindle speed of the tool is not exceeded, you can program a limitation. (Specify it in the **Nmax** column of the "tool.t" tool table.)

Notes on programming

- The values entered for the module, number of teeth and outside diameter (outside diameter) are monitored. If these values are not coherent, then an error message is displayed. You can fill in 2 of the 3 parameters. Enter 0 for the module, the number of teeth, or the outside diameter (outside diameter). In this case, the control will calculate the missing value.
- Program FUNCTION TURNDATA SPIN VCONST:OFF.
- If you program FUNCTION TURNDATA SPIN VCONST:OFF S15, then the spindle speed of the tool is calculated as follows: **Q541** x S. With **Q541**=238 and S=15, this would result in a tool spindle speed of 3570 rpm.
- Program the direction of rotation of your workpiece (**M303**/**M304**) before the start of the cycle.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2/3)? Define extent of machining: **0**: Roughing and finishing **1**: Only roughing **2**: Only finishing to final dimension **3**: Only finishing to oversize Input: **0**, **1**, **2**, **3**
	Q540 Module? Module of the gear Input: **0...99.999**
	Q541 Number of teeth? Describe gear: number of teeth Input: **0...99999**
	**Q542 Outside diameter?** Describe gear: outside diameter of finished part Input: **0...99999.9999**
	Q543 Trough-to-tip clearance? Distance between the addendum circle of the gear to be made and root circle of the mating gear. Input: **0...9.9999**
	Q544 Angle of inclination? Angle at which the teeth of a helical gear are inclined relative to the direction of the axis. For straight-cut gears, this angle is 0°. Input: **-60...+60**
	Q545 Tool lead angle? Angle of the edges of the gear hob. Enter this value in decimal notation. Example: 0°47'=0.7833 Input: **-60...+60**
	Q546 Reverse tool rotation direction? Describe tool: Direction of spindle rotation of the gear hob **3**: Clockwise rotating tool (**M3**) **4**: Counterclockwise rotating tool (**M4**) Input: **3**, **4**
	Q547 Angle offset of tool spindle? Angle at which the control turns the workpiece at the beginning of the cycle. Input: **-180...+180**

Help graphic	Parameter
	Q550 Machining side (0=pos./1=neg.)? Define at which side machining is to take place. **0**: Positive machining side of the main axis in the I-CS **1**: Negative machining side of the main axis in the I-CS Input: **0**, **1**
	Q533 Preferred dir. of incid. angle? Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on your machine. In general, there are always two possible solutions. Via parameter **Q533**, you configure which solution option the control is to use: **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **1**: Automatically position the tilting axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (**TURN**) Input: **1**, **2**
	Q253 Feed rate for pre-positioning? Definition of the traversing speed of the tool during tilting and during pre-positioning. And during positioning of the tool axis between the individual infeeds. Feed rate is in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q553 TOOL:L offset, machining start? Define the minimum length offset (L OFFSET) that the tool should have when in use. The control offsets the tool in the longitudinal direction by this amount. This value has an incremental effect. Input: **0...999.999**

Help graphic	Parameter
	Q551 Starting point in Z? Starting point of the hobbing process in Z Input: **-99999.9999...+99999.9999**
	Q552 End point in Z? End point of the hobbing process in Z Input: **-99999.9999...+99999.9999**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0,001...999.999**
	Q460 Set-up clearance? Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...999.999**
	Q488 Feed rate for plunging Feed rate of the tool infeed Input: **0...99999.999** or **FAUTO**
	Q478 Roughing feed rate? Freed rate during roughing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. If M136 has been programmed, the value is interpreted by the control in millimeters per revolution; without M136, in millimeters per minute. Input: **0...99999.999** or **FAUTO**

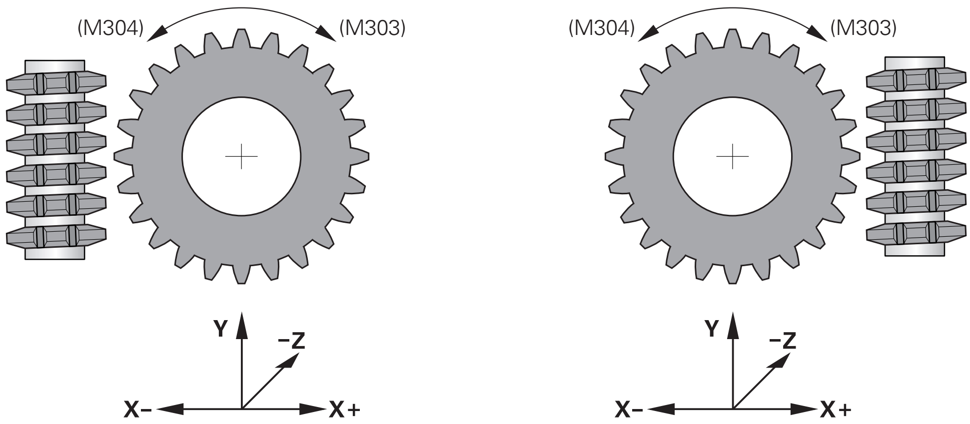
Example

```
11 CYCL DEF 880 GEAR HOBBING ~
    Q215=+0                  ;MACHINING OPERATION ~
    Q540=+0                  ;MODULE ~
    Q541=+0                  ;NUMBER OF TEETH ~
    Q542=+0                  ;OUTSIDE DIAMETER ~
    Q543=+0.1666             ;TROUGH-TIP CLEARANCE ~
    Q544=+0                  ;ANGLE OF INCLINATION ~
    Q545=+0                  ;TOOL LEAD ANGLE ~
    Q546=+3                  ;CHANGE TOOL DIRECTN. ~
    Q547=+0                  ;ANG. OFFSET, SPINDLE ~
    Q550=+1                  ;MACHINING SIDE ~
    Q533=+0                  ;PREFERRED DIRECTION ~
    Q530=+2                  ;INCLINED MACHINING ~
    Q253=+750                ;F PRE-POSITIONING ~
    Q260=+100                ;CLEARANCE HEIGHT ~
    Q553=+10                 ;TOOL LENGTH OFFSET ~
    Q551=+0                  ;STARTING POINT IN Z
    Q552=-10                 ;END POINT IN Z
    Q463=+1                  ;MAX. CUTTING DEPTH ~
    Q460=+2                  ;SAFETY CLEARANCE ~
    Q488=+0.3                ;PLUNGING FEED RATE ~
    Q478=+0.3                ;ROUGHING FEED RATE ~
    Q483=+0.4                ;OVERSIZE FOR DIAMETER ~
    Q505=+0.2                ;FINISHING FEED RATE
```

Direction of rotation depending on the machining side (Q550)

Determine the direction of rotation of the rotary table:

1 **What tool? (Right-cutting/left-cutting?)**
2 **What machining side? X+ (Q550=0) / X- (Q550=1)**
3 **Look up the direction of rotation of the rotary table in one of the two tables below!** To do so, select the appropriate table for the direction of rotation of your tool (**right-cutting/left-cutting**). Please refer to the tables below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)** ab.



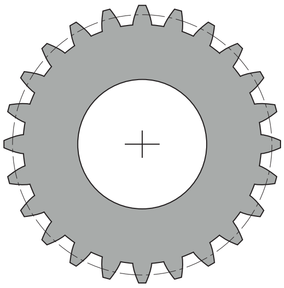
Tool: Right-cutting M3	
Machining side X+ (Q550=0)	Direction of rotation of the table: Clockwise (M303)
Machining side X- (Q550=1)	Direction of rotation of the table: Counterclockwise (M304)

Tool: Left-cutting M4	
Machining side X+ (Q550=0)	Direction of rotation of the table: Counterclockwise (M304)
Machining side X- (Q550=1)	Direction of rotation of the table: Clockwise (M303)

14.6.3 Gear manufacturing fundamentals (option 157)

Fundamentals

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



For the cycles, option 157 Gear Cutting is required. If you would like to use these cycles in turning mode, you also need option 50. In milling mode, the tool spindle is the master spindle, in turning mode, it is the workpiece spindle. The other spindle is called slave spindle. Depending on the operating mode, you program the speed or the cutting speed with a **TOOL CALL S** or **FUNCTION TURNDATA SPIN**.

To orient the I-CS coordinate system, Cycles **286** and **287** use the precession angle that is also affected by Cycles **800** and **801** in turning mode. At the end of the cycle, the control resets the precession angle to its state at the beginning of the cycle. If one of these cycles is aborted, the precession angle will also be reset.

The axis crossing angle is the angle between workpiece and tool. It results from the angle of inclination of the tool and the angle of inclination of the gear. Based on the required axis crossing angle, Cycles **286** and **287** calculate the required inclination of the rotary axis at the machine. The cycles will always position the first rotary axis starting from the tool.

In order to ensure that the tool can safely be retracted from the gear in the event of a fault (NC stop or power failure), the cycles automatically control the **LiftOff**. The cycles define the direction and path for a **LiftOff**.

The gear itself will first be described in Cycle **285 DEFINE GEAR**. Then, program Cycle **286 GEAR HOBBING** or Cycle **287 GEAR SKIVING**.

Program the following:

- Call a tool with **TOOL CALL**
- Select turning mode or milling mode, with **FUNCTION MODE TURN** or **FUNCTION MODE MILL** "**KINEMATIC_GEAR**" kinematics selection
- Spindle direction of rotation, e.g. **M3** or **M303**
- Perform pre-positioning for the cycle depending on your selection of **MILL** or **TURN**
- Define the **CYCL DEF 285 DEFINE GEAR** cycle
- Define the **CYCL DEF 286 GEAR HOBBING** or **CYCL DEF 287 GEAR SKIVING** cycle.

Notes

NOTICE

Danger of collision!

If you do not pre-position the tool to a safe position, a collision between tool and workpiece (fixtures) may occur during tilting.

- Pre-position the tool to a safe position

NOTICE

Danger of collision!

If the workpiece is clamped too deeply into the fixture, a collision between tool and fixture might occur during machining. The starting point in Z and the end point in Z are extended by the set-up clearance **Q200**!

- Make sure to clamp the workpiece in such a way that it projects far enough from the fixture and no collision can occur between tool and fixture.

- Before calling the cycle, set the preset to the center of rotation of the workpiece spindle.
- Please note that the slave spindle will continue to rotate after the end of the cycle. If you want to stop the spindle before the end of the program, make sure to program a corresponding M function.
- Activate the **LiftOff** in the tool table. In addition, this function must have been configured by your machine manufacturer.
- Remember that you need to program the speed of the master spindle before calling the cycle, i.e. the tool spindle speed in milling mode and the workpiece spindle speed in turning mode.

Gear formulas

Speed calculation

- n_T: Tool spindle speed
- n_W: Workpiece spindle speed
- z_T: Number of tool teeth
- z_W: Number of workpiece teeth

Definition	Tool spindle	Workpiece spindle
Hobbing	$n_T = n_W * z_w$	$n_W = \frac{n_T}{z_W}$
Skiving	$n_T = n_W * \frac{z_W}{z_T}$	$n_W = n_T * \frac{z_T}{z_W}$

Straight-cut spur gears

- m: Module (**Q540**)
- p: Pitch
- h: Tooth height (**Q563**)
- d: Pitch-circle diameter
- z: Number of teeth (**Q541**)
- c: Trough-to-tip clearance (**Q543**)
- d_a: Diameter of the addendum circle (outside diameter, **Q542**)
- d_f: Root circle diameter

Definition	Formula
Module (**Q540**)	$m = \frac{p}{\pi}$ $m = \frac{d}{z}$
Pitch	$p = \pi * m$
Pitch-circle diameter	$d = m * z$
Tooth height (**Q563**)	$h = 2 * m + c$
Diameter of the addendum circle (outside diameter, **Q542**)	$d_a = m * (z + 2)$ $d_a = d + 2 * m$
Root circle diameter	$d_f = d - 2 * (m + c)$
Root circle diameter if tooth height > 0	$d_f = d_a - 2 * (h + c)$
Number of teeth (**Q541**)	$z = \frac{d}{m}$ $z = \frac{d_a - 2 * m}{m}$

Remember to observe the algebraic sign when calculating an inner gear.

Example: Calculating the diameter of the addendum circle (outside diameter)

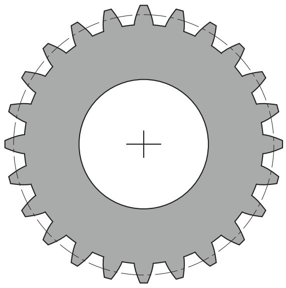
Outer gear: **Q540** * (**Q541** + 2) = 1 * (+46 + 2)

Inner gear: **Q540** * (**Q541** + 2) = 1 * (-46 + 2)

14.6.4 Cycle 285 DEFINE GEAR (option 157)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



Use Cycle **285 DEFINE GEAR** to describe the geometry of the gearing system. To describe the tool, use Cycle **286 GEAR HOBBING** or Cycle **287GEAR SKIVING** and the tool table (TOOL.T).

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- This cycle is DEF-active. The values of these Q parameters will only be read when a CALL-active machining cycle is executed. If you overwrite these input parameters after the cycle definition and before calling the machining cycle, the gear geometry will be modified.
- Define the tool as a milling cutter in the tool table.

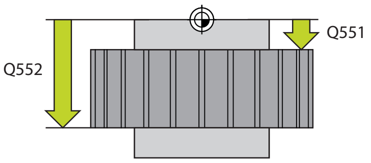
Notes on programming

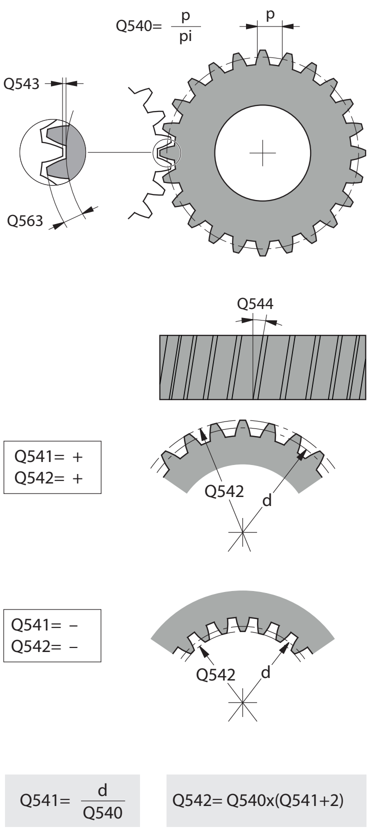
- You must specify values for module and number of teeth. If the outside diameter (diameter of the addendum circle) and the tooth height are defined as 0, normal running gears (DIN 3960) will be machined. If you want to machine gearing systems that differ from this standard, define the corresponding geometry by specifying the diameter of the addendum circle (outside diameter) **Q542** and the tooth height **Q563**.
- If the algebraic signs of the two input parameters **Q541** and **Q542** are contradictory, the cycle will be aborted with an error message.
- Remember that the diameter of the addendum circle is always greater than the root circle diameter, even for an inner gear.

  Inner gear example: The outside diameter (addendum circle) is –40 mm, the root circle diameter is –45 mm. Also in this case, the diameter of the addendum circle (outside diameter) is (numerically) greater than the root circle diameter.

Cycle parameters

Help graphic





Parameter

Q551 Starting point in Z?
Starting point of the hobbing process in Z
Input: **-99999.9999...+99999.9999**

Q552 End point in Z?
End point of the hobbing process in Z
Input: **-99999.9999...+99999.9999**

Q540 Module?
Module of the gear
Input: **0...99.999**

Q541 Number of teeth?
Number of teeth. This parameter depends on **Q542**.

+: If the number of teeth is positive, and at the same time the parameter **Q542** is positive, then an external gear will be machined.

-: If the number of teeth is negative, and at the same time the parameter **Q542** is negative, then an internal gear will be machined.

Input: **-99999...+99999**

Q542 Outside diameter?
Addendum circle (outside diameter) of the gear. This parameter depends on **Q541**.

+: If the addendum circle is positive, and at the same time the parameter **Q541** is positive, then an external gear will be machined.

–: If the addendum circle is negative, and at the same time the parameter **Q541** is negative, then an internal gear will be machined.

Input: **-9999.9999...+9999.9999**

Q563 Tooth height?
Distance from the tooth trough to the tooth tip.
Input: **0...999.999**

Q543 Trough-to-tip clearance?
Distance between the addendum circle of the gear to be made and root circle of the mating gear.
Input: **0...9.9999**

Q544 Angle of inclination?
Angle at which the teeth of a helical gear are inclined relative to the direction of the axis. For straight-cut gears, this angle is 0°.
Input: **-60...+60**

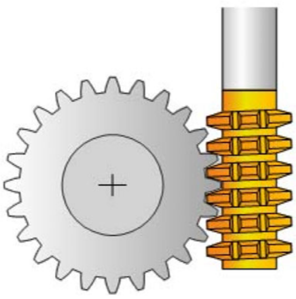
Example

11 CYCL DEF 285 DEFINE GEAR ~	
Q551=+0	;STARTING POINT IN Z ~
Q552=-10	;END POINT IN Z ~
Q540=+1	;MODULE ~
Q541=+10	;NUMBER OF TEETH ~
Q542=+0	;OUTSIDE DIAMETER ~
Q563=+0	;TOOTH HEIGHT ~
Q543=+0.17	;TROUGH-TIP CLEARANCE ~
Q544=+0	;ANGLE OF INCLINATION

14.6.5 Cycle 286 GEAR HOBBING (option 157)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



With Cycle **286 GEAR HOBBING**, you can machine external cylindrical gears or helical gears with any angles. You can select the machining strategy and the machining side in the cycle. The machining process for gear hobbing is performed with a synchronized rotary movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction. Both for roughing and for finishing, the cutting operation may be offset by x edges relative to a height defined at the tool (e.g. 10 cutting edges for a height of 10 mm). This means that all cutting edges will be used in order to increase the tool life of the tool.

Cycle sequence

1. The control positions the tool in the tool axis to clearance height **Q260** at the feed rate **FMAX**. If the tool is already at a location in the tool axis higher than **Q260**, the tool will not be moved.
2. Before tilting the working plane, the control positions the tool in X to a safe coordinate at the **FMAX** feed rate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3. The control then tilts the working plane at the feed rate **Q253**
4. The control positions the tool at the feed rate **FMAX** to the starting point in the working plane
5. The control then moves the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q200**.
6. The control moves the tool at the defined feed rate **Q478** (for roughing) or **Q505** (for finishing) to hob the workpiece in longitudinal direction. The area to be machined is limited by the starting point in Z **Q551**+**Q200** and by the end point in Z **Q552**+**Q200** (**Q551** and **Q552** are defined in Cycle **285**).

   Further information: "Cycle 285 DEFINE GEAR (option 157)", Page 909
7. When the tool reaches the end point, it is retracted at the feed rate **Q253** and returns to the starting point.
8. The control repeats the steps 5 to 7 until the defined gear is completed.
9. Finally, the control retracts the tool to the clearance height **Q260** at the feed rate **FMAX**.

Notes

NOTICE

Danger of collision!

When programming helical gears, the rotary axes will remain tilted, even after the end of the program. There is a danger of collision!

- Make sure to retract the tool before changing the position of the tilting axis

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The cycle is CALL-active.
- The maximum speed of the rotary table cannot be exceeded. If you have specified a higher value under **NMAX** in the tool table, the control will decrease the value to the maximum speed.

Avoid master spindle speeds of less than 6 rpm. Otherwise, it is not possible to reliably use a feed rate in mm/rev.

Notes on programming

- In order to ensure constant engagement of the cutting edge of a tool, you need to define a very small path in cycle parameter **Q554 SYNCHRONOUS SHIFT**.
- Make sure to program the direction of rotation of the master spindle (channel spindle) before the cycle start.
- If you program **FUNCTION TURNDATA SPIN VCONST:OFF S15**, the spindle speed of the tool is calculated as **Q541** x S. With **Q541** = 238 and S = 15, this would result in a tool spindle speed of 3570 rpm.

Cycle parameters

Help graphic	Parameter

Q215 Machining operation (0/1/2/3)?

Define extent of machining:

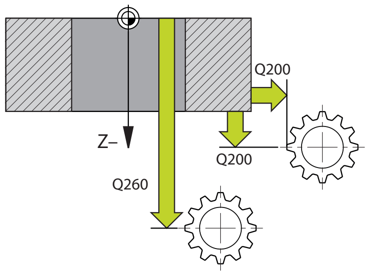
0: Roughing and finishing

1: Only roughing

2: Only finishing to final dimension

3: Only finishing to oversize

Input: **0**, **1**, **2**, **3**



Q200 Set-up clearance?

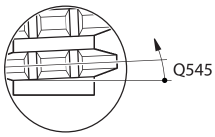
Distance for retraction and prepositioning. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

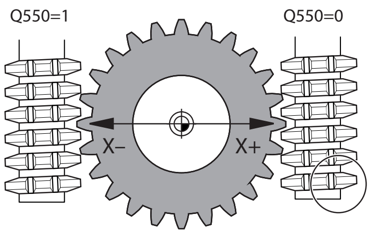


Q545 Tool lead angle?

Angle of the edges of the gear hob. Enter this value in decimal notation.

Example: 0°47'=0.7833

Input: **-60...+60**



Q546 Reverse spindle rotation dir.?

Direction of rotation of the slave spindle:

0: No change in the direction of rotation

1: Change in the direction of rotation

Input: **0**, **1**

Page 917

Q547 Angle offset of tool spindle?

Angle at which the control turns the workpiece at the beginning of the cycle.

Input: **-180...+180**

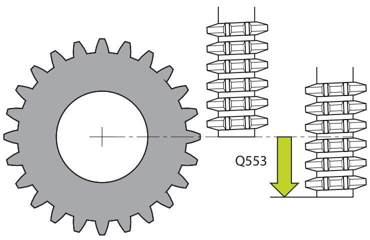
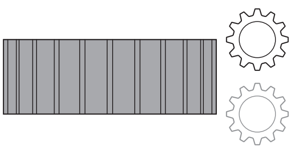
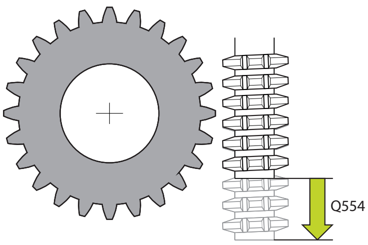
Q550 Machining side (0=pos./1=neg.)?

Define at which side machining is to take place.

0: Positive machining side of the main axis in the I-CS

1: Negative machining side of the main axis in the I-CS

Input: **0**, **1**

Help graphic	Parameter
	Q533 Preferred dir. of incid. angle? Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on your machine. In general, there are always two possible solutions. Via parameter **Q533**, you configure which solution option the control is to use: **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **1**: Automatically position the tilting axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (**TURN**) Input: **1**, **2**
	**Q253 Feed rate for pre-positioning?** Definition of the traversing speed of the tool during tilting and during pre-positioning. And during positioning of the tool axis between the individual infeeds. Feed rate is in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	**Q553 TOOL:L offset, machining start?** Define the minimum length offset (L OFFSET) that the tool should have when in use. The control offsets the tool in the longitudinal direction by this amount. This value has an incremental effect. Input: **0...999.999**
	**Q554 Path for synchronous shift?** Define by which distance the gear hob will be offset in its axial direction during machining. This way, tool wear can be distributed over this area of the cutting edges. For helical gears, it is thus possible to limit the cutting edges used for machining. Entering **0** deactivates the synchronous shift function. Input: **-99...+99.9999**

Help graphic	Parameter
	Q548 Tool shift for roughing? Specify the number of cutting edges by which the control will shift the roughing tool in its axial direction. The shift will be performed incrementally relative to parameter **Q553**. Entering 0 deactivates the shift function. Input: **-99...+99**
	Q463 Maximum cutting depth? Maximum infeed (radius value) in the radial direction. The infeed is distributed evenly to avoid abrasive cuts. Input: **0,001...999.999**
	Q488 Feed rate for plunging Feed rate for tool infeed. The control interprets the feed rate in mm per workpiece revolution. Input: **0...99999.999** or **FAUTO**
	Q478 Roughing feed rate? Feed rate during roughing. The control interprets the feed rate in mm per workpiece revolution. Input: **0...99999.999** or **FAUTO**
	Q483 Oversize for diameter? Diameter oversize on the defined contour. This value has an incremental effect. Input: **0...99.999**
	Q505 Finishing feed rate? Feed rate during finishing. The control interprets the feed rate in mm per workpiece revolution. Input: **0...99999.999** or **FAUTO**
	Q549 Tool shift for finishing? Specify the number of cutting edges by which the control will shift the finishing tool in its longitudinal direction. The shift will be performed incrementally relative to parameter **Q553**. Entering 0 deactivates the shift function. Input: **-99...+99**

Example

```
11 CYCL DEF 286 GEAR HOBBING ~
    Q215=+0              ;MACHINING OPERATION ~
    Q200=+2              ;SET-UP CLEARANCE ~
    Q260=+100            ;CLEARANCE HEIGHT ~
    Q545=+0              ;TOOL LEAD ANGLE ~
    Q546=+0              ;CHANGE ROTATION DIR. ~
    Q547=+0              ;ANG. OFFSET, SPINDLE ~
    Q550=+1              ;MACHINING SIDE ~
    Q533=+0              ;PREFERRED DIRECTION ~
    Q530=+2              ;INCLINED MACHINING ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q553=+10             ;TOOL LENGTH OFFSET ~
    Q554=+0              ;SYNCHRONOUS SHIFT ~
    Q548=+0              ;ROUGHING SHIFT ~
    Q463=+1              ;MAX. CUTTING DEPTH ~
    Q488=+0.3            ;PLUNGING FEED RATE ~
    Q478=+0.3            ;ROUGHING FEED RATE ~
    Q483=+0.4            ;OVERSIZE FOR DIAMETER ~
    Q505=+0.2            ;FINISHING FEED RATE ~
    Q549=+0              ;FINISHING SHIFT
```

Verifying and changing directions of rotation of the spindles

Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

Determine the direction of rotation of the rotary table:

1 What tool? (Right-cutting/left-cutting?)
2 Which machining side? **X+ (Q550=0) / X- (Q550=1)**
3 Look up the direction of rotation of the rotary table in one of the two tables below! To do so, select the appropriate table for the direction of rotation of your tool (right-cutting/left-cutting). Please refer to the appropriate table below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)**.

Tool: Right-cutting M3

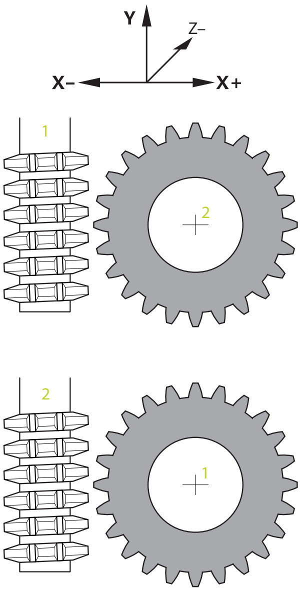
Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Clockwise.)(e. g. **M303**)
X- (Q550=1)	Counterclockwise (e. g. **M304**)

Tool: Left-cutting M4

Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Counterclockwise (e. g. **M304**)
X- (Q550=1)	Clockwise.)(e. g. **M303**)

Keep in mind that in special cases, the directions of rotation might deviate from the ones indicated in these tables.

Changing the direction of rotation



Milling:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

Turning:

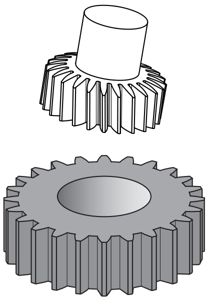
- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

> Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.
>
> If required, define a low spindle speed to make sure that the direction of rotation is correct.

14.6.6 Cycle 287 GEAR SKIVING option 157

Application

	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



With Cycle **287 GEAR SKIVING**, you can machine cylindrical gears or helical gears with any angles. Cutting takes place on the one hand by the axial feeding of the tool and on the other hand through the rolling motion.

You can select the machining side in the cycle. The machining process for gear skiving is performed with a synchronized rotary movement of the tool spindle and workpiece spindle. In addition, the cutter moves along the workpiece in axial direction.

In the cycle, you can call a table containing technology data. In this table, you can define the feed rate, the lateral infeed, and the lateral offset for each cut.

Further information: "Technology table for Cycle 287 Gear Skiving", Page 1878

Cycle sequence

1 The control positions the tool in the tool axis to the clearance height **Q260** at the feed rate **FMAX**. If the value of the current tool position in the tool axis is greater than **Q260**, the tool will not be moved
2 Before tilting the working plane, the control positions the tool in X at the feed rate **FMAX** to a safe coordinate. If the tool is already located at a coordinate in the working plane that is greater than the calculated coordinate, the tool is not moved.
3 The control tilts the working plane at the feed rate **Q253**
4 The control positions the tool at the feed rate **FMAX** to the starting point in the working plane
5 After that, the control positions the tool in the tool axis at the feed rate **Q253** to the set-up clearance **Q200**
6 The control then traverses the approach length. The control automatically calculates this distance. The approach length is the distance from the initial scratch to the complete plunging depth.
7 The control rolls the tool over the workpiece to be geared in longitudinal direction at the defined feed rate. During the initial infeed of the cut **Q586** the control moves at the initial feed rate **Q588**. The control then uses intermediate values for the infeed and feed rate of the next cuts. The control calculates these values itself. The intermediate feed rate values, however, depend on the factor for feed-rate adaptation **Q580**. When the control arrives at the last infeed **Q587**, it performs the last cut with the feed rate **Q589**

8 The area to be machined is limited by the starting point in Z **Q551**+**Q200** and by the end point in Z **Q552** (**Q551** and **Q552** are defined in Cycle **285**). The approach length must be added to the starting point. Its purpose is to prevent the tool from plunging into the workpiece all the way to the machining diameter. The control calculates this distance itself.
9 At the end of machining, the tool moves beyond the defined end point by the overrun path **Q580**. The overrun path serves to completely machine the gear.
10 When the control reaches the end point, it retracts the tool at the feed rate **Q253** and positions it back to the starting point
11 Finally the control positions the tool to the clearance height **Q260** at the feed rate FMAX

Notes

NOTICE

Danger of collision!

When programming helical gears, the rotary axes will remain tilted, even after the end of the program. There is a danger of collision!

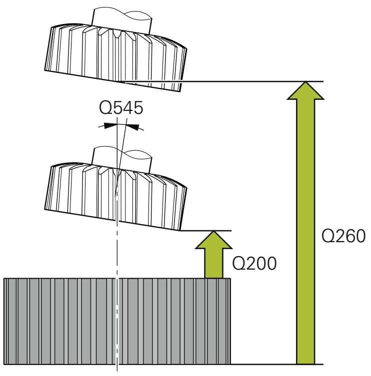
▶ Make sure to retract the tool before changing the position of the tilting axis

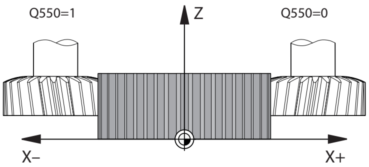
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The cycle is CALL-active.
- The speed ratio between tool and workpiece results from the number of teeth of the gear wheel and the number of cutting edges of the tool.

Notes on programming

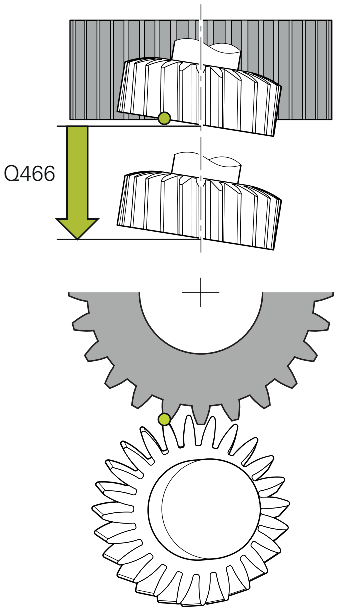
- Make sure to program the direction of rotation of the master spindle (channel spindle) before the cycle start.
- The larger the factor in **Q580 FEED-RATE ADAPTION**, the earlier the control will adapt the feed rate to the feed rate for the last cut. The recommended value is 0.2.
- When defining the tool, make sure to specify the number of cutting edges as indicated in the tool table.
- If only two cuts have been programmed in **Q240**, the last infeed from **Q587** and the last feed rate from **Q589** will be ignored. If only one cut has been programmed, the first infeed from **Q586** will also be ignored.

Cycle parameters

Help graphic	Parameter
	Q240 Number of cuts? Number of cuts to the final depth **0**: The control automatically determines the minimum number of cuts **1**: One cut **2**: Two cuts where the control considers only the infeed for the first cut **Q586**. The control does not consider the infeed for the last cut **Q587**. **3 to 99**: Programmed number of cuts **"..."**: Path of a table containing technology data see "Technology table for Cycle 287 Gear Skiving", Page 1878 Input: **0...99** or text entry of max. **255** characters or **QS** parameter
	Q584 Number of the first cut? Define which cut number the control will perform first. Input: **1...999**
	Q585 Number of the last cut? Define at which number the control will perform the last cut. Input: **1...999**
 	**Q200 Set-up clearance?** Distance for retraction and prepositioning. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis in which no collision with the workpiece can occur (for intermediary positioning and retraction at the end of the cycle). The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q545 Tool lead angle? Angle of the edges of the skiving tool. Enter this value in decimal notation. Example: 0°47' = 0.7833 Input: **-60...+60**
	Q546 Reverse spindle rotation dir.? Direction of rotation of the slave spindle: **0**: No change in the direction of rotation **1**: Change in the direction of rotation Input: **0**, **1** Page 925
	Q547 Angle offset of tool spindle? Angle at which the control turns the workpiece at the beginning of the cycle. Input: **-180...+180**

Help graphic	Parameter
	**Q550 Machining side (0=pos./1=neg.)?** Define at which side machining is to take place. **0**: Positive machining side of the main axis in the I-CS **1**: Negative machining side of the main axis in the I-CS Input: **0**, **1**
	Q533 Preferred dir. of incid. angle? Selection of alternate possibilities of inclination. The angle of incidence you define is used by the control to calculate the appropriate positioning of the tilting axes present on your machine. In general, there are always two possible solutions. Via parameter **Q533**, you configure which solution option the control is to use: **0**: Solution that is the shortest distance from the current position **-1**: Solution that is in the range between 0° and -179.9999° **+1**: Solution that is in the range between 0° and +180° **-2**: Solution that is in the range between -90° and -179.9999° **+2**: Solution that is between +90° and +180° Input: **-2**, **-1**, **0**, **+1**, **+2**
	Q530 Inclined machining? Position the tilting axes for inclined machining: **1**: Automatically position the tilting axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and tool remains unchanged. The control performs a compensating movement with the linear axes **2**: Automatically position the tilting axis without orienting the tool tip (**TURN**) Input: **1**, **2**
	Q253 Feed rate for pre-positioning? Definition of the traversing speed of the tool during tilting and during pre-positioning. And during positioning of the tool axis between the individual infeeds. Feed rate is in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q586 Infeed for first cut? Infeed for the first cut. This value has an incremental effect. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 1878 Input: **0,001...99.999**
	Q587 Infeed for last cut? Infeed for the last cut. This value has an incremental effect. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 1878 Input: **0,001...99.999**

Help graphic	Parameter
	Q588 Feed rate for first cut? Feed rate for the first cut. The control interprets the feed rate in mm per workpiece revolution. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 1878 Input: **0,001...99.999**
	Q589 Feed rate for last cut? Feed rate for the last cut. The control interprets the feed rate in mm per workpiece revolution. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 1878 Input: **0,001...99.999**
	Q580 Factor for feed-rate adaptation? Using this factor, you can define a feed rate reduction. This is due to the fact that the feed rate must decrease with increasing cutting numbers. The greater the value, the earlier the control will adapt the feed rates to match the last feed rate. If the path of a technology table is stored in **Q240**, this parameter has no effect. see "Technology table for Cycle 287 Gear Skiving", Page 1878 Input: **0...1**

Help graphic	Parameter
	**Q466 Overrun path?** Length of overtravel at the end of the gear teeth. The overtravel path ensures that the control machines the gear teeth up to the desired end point. If you do not program these optional parameters, then the control uses the safety clearance **Q200** as the overtravel path. Input: **0.1...99.9**

Example

11 CYCL DEF 287 GEAR SKIVING ~	
Q240=+0	;NUMBER OF CUTS ~
Q584=+1	;NO. OF FIRST CUT ~
Q585=+999	;NO. OF LAST CUT ~
Q200=+2	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q545=+0	;TOOL LEAD ANGLE ~
Q546=+0	;CHANGE ROTATION DIR. ~
Q547=+0	;ANG. OFFSET, SPINDLE ~
Q550=+1	;MACHINING SIDE ~
Q533=+0	;PREFERRED DIRECTION ~
Q530=+2	;INCLINED MACHINING ~
Q253=+750	;F PRE-POSITIONING ~
Q586=+1	;FIRST INFEED ~
Q587=+0.1	;LAST INFEED ~
Q588=+0.2	;FIRST FEED RATE ~
Q589=+0.05	;LAST FEED RATE ~
Q580=+0.2	;FEED-RATE ADAPTION ~
Q466=+2	;OVERRUN PATH

Verifying and changing directions of rotation of the spindles

Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.

Determine the direction of rotation of the rotary table:

1 What tool? (Right-cutting/left-cutting?)
2 Which machining side? **X+ (Q550=0) / X- (Q550=1)**
3 Look up the direction of rotation of the rotary table in one of the two tables below! To do so, select the appropriate table for the direction of rotation of your tool (right-cutting/left-cutting). Please refer to the appropriate table below to find the direction of rotation of your rotary table for the desired machining side **X+ (Q550=0) / X- (Q550=1)**.

Tool: Right-cutting M3

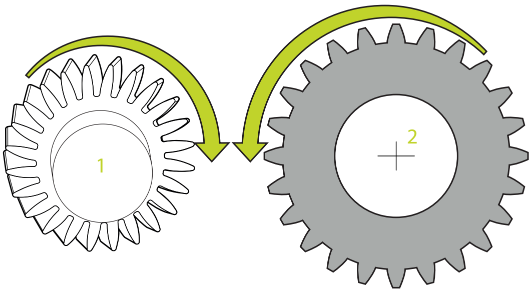
Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Clockwise. (e. g. **M303**)
X- (Q550=1)	Counterclockwise (e. g. **M304**)

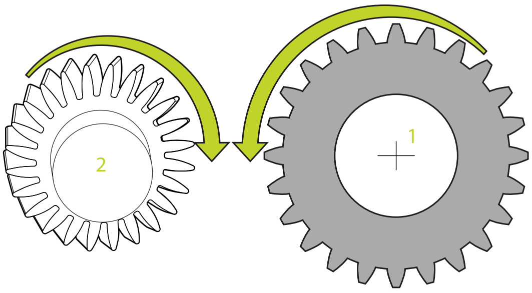
Tool: Left-cutting M4

Machining side	Direction of rotation of the rotary table
X+ (Q550=0)	Counterclockwise (e. g. **M304**)
X- (Q550=1)	Clockwise. (e. g. **M303**)

Keep in mind that in special cases, the directions of rotation might deviate from the ones indicated in these tables.

Changing the direction of rotation





Milling:

- Master spindle **1**: Use M3 or M4 to define the tool spindle as the master spindle. This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

Turning:

- Master spindle **1**: Use an M function to define the tool spindle as the master spindle. This M function is machine manufacturer-specific (M303, M304,...). This defines the direction of rotation (changing the direction of rotation of the master spindle does not affect the direction of rotation of the slave spindle)
- Slave spindle **2**: To change the direction of rotation of the slave spindle, adjust the value of input parameter **Q546**.

> Before performing a machining operation, make sure that the direction of rotation has been set correctly for both spindles.
>
> If required, define a low spindle speed to make sure that the direction of rotation is correct.

14.6.7 Programming examples

Example: Gear hobbing

The following NC program uses Cycle **880 GEAR HOBBING** This programming example illustrates the machining of a helical gear, with Module=2.1.

Program sequence

- Tool call: Gear hob
- Start turning mode
- Move to safe position
- Call the cycle
- Reset the coordinate system with Cycle 801 and M145

0 BEGIN PGM 8 MM	
1 BLK FORM CYLINDER Z R42 L150	
2 FUNCTION MODE MILL	; activate milling mode
3 TOOL CALL "GEAD_HOB"	; call tool
4 FUNCTION MODE TURN	; activate turning mode
5 CYCL DEF 801 KOORDINATEN-SYSTEM ZURUECKSETZEN	
6 M145	; cancel a potentially still active M144
7 FUNCTION TURNDATA SPIN VCONST:OFF S50	; constant cutting speed OFF
8 M140 MB MAX	; retract the tool
9 L A+0 R0 FMAX	; set turning axis to 0
10 L X+250 Y-250 R0 FMAX M303	; pre-position the tool in the working plane on the side on which machining will be performed, Spindle ON
11 L Z+20 R0 FMAX	; pre-position the tool in the spindle axis
12 M136	; feed rate in mm/rev.
13 CYCL DEF 880 GEAR HOBBING ~	
Q215=+0 ;MACHINING OPERATION ~	
Q540=+2.1 ;MODULE ~	
Q541=+0 ;NUMBER OF TEETH ~	
Q542=+69.3 ;OUTSIDE DIAMETER ~	
Q543=+0.1666 ;TROUGH-TIP CLEARANCE ~	
Q544=-5 ;ANGLE OF INCLINATION ~	
Q545=+1.6833 ;TOOL LEAD ANGLE ~	
Q546=+3 ;CHANGE TOOL DIRECTN. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+0 ;MACHINING SIDE ~	
Q533=+0 ;PREFERRED DIRECTION ~	
Q530=+2 ;INCLINED MACHINING ~	
Q253=+800 ;F PRE-POSITIONING ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q553=+10 ;TOOL LENGTH OFFSET ~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-10 ;END POINT IN Z ~	

Q463=+1 ;MAX. CUTTING DEPTH ~	
Q460=2 ;SAFETY CLEARANCE ~	
Q488=+1 ;PLUNGING FEED RATE ~	
Q478=+2 ;ROUGHING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q505=+1 ;FINISHING FEED RATE	
14 CYCL CALL	; call cycle
15 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
16 M145	; switch off active M144 in the cycle
17 FUNCTION MODE MILL	; activate milling mode
18 M140 MB MAX	; retract tool in the tool axis
19 L A+0 C+0 R0 FMAX	; reset turning
20 M30	; end of program
21 END PGM 8 MM	

Example of hob milling

The following NC program uses Cycle **286 GEAR HOBBING**. This programming example shows how to machine an involute spline with module = 1 (deviating from DIN 3960).

Program sequence

- Tool call: Gear hob
- Start the turning mode
- Reset the coordinate system with Cycle **801**
- Move to safe position
- Define Cycle **285**
- Call Cycle **286**
- Reset the coordinate system with Cycle **801**

0 BEGIN PGM 7 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI58	
2 TOOL CALL "GEAR_HOB"	; Call the tool
3 FUNCTION MODE TURN	; Activate turning mode
* - ...	; Reset the coordinate system
4 CYCL DEF 801 KOORDINATEN-SYSTEM ZURUECKSETZEN	
5 M145	; Cancel a potentially still active M144
6 FUNCTION TURNDATA SPIN VCONST:OFF S50	; Constant surface speed OFF
7 M140 MB MAX	; Retract the tool
8 L A+0 R0 FMAX	; Set the rotary axis to 0
9 L X+0 Y+0 R0 FMAX	; Pre-position the tool at the workpiece center
10 L Z+50 R0 FMAX	; Pre-position the tool in the spindle axis
11 CYCL DEF 285 DEFINE GEAR ~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-11 ;END POINT IN Z ~	
Q540=+1 ;MODULE ~	
Q541=+90 ;NUMBER OF TEETH ~	
Q542=+90 ;OUTSIDE DIAMETER ~	
Q563=+1 ;TOOTH HEIGHT ~	
Q543=+0.05 ;TROUGH-TIP CLEARANCE ~	
Q544=-10 ;ANGLE OF INCLINATION	
12 CYCL DEF 286 GEAR HOBBING ~	
Q215=+0 ;MACHINING OPERATION ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q260=+30 ;CLEARANCE HEIGHT ~	
Q545=+1.6 ;TOOL LEAD ANGLE ~	
Q546=+0 ;CHANGE ROTATION DIR. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+1 ;MACHINING SIDE ~	
Q533=+1 ;PREFERRED DIRECTION ~	
Q530=+2 ;INCLINED MACHINING ~	

Q253=+2222 ;F PRE-POSITIONING ~	
Q553=+5 ;TOOL LENGTH OFFSET ~	
Q554=+10 ;SYNCHRONOUS SHIFT ~	
Q548=+1 ;ROUGHING SHIFT ~	
Q463=+1 ;MAX. CUTTING DEPTH ~	
Q488=+0.3 ;PLUNGING FEED RATE ~	
Q478=+0.3 ;PLUNGING FEED RATE ~	
Q483=+0.4 ;OVERSIZE FOR DIAMETER ~	
Q505=+0.2 ;FINISHING FEED RATE ~	
Q549=+3 ;FINISHING SHIFT	
13 CYCL CALL M303	; Call the cycle, spindle ON
14 FUNCTION MODE MILL	; Activate milling mode
15 M140 MB MAX	; Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	; Reset the rotation
17 M30	; End of program
18 END PGM 7 MM	

Example of skiving

The following NC program uses Cycle **287 GEAR SKIVING**. This programming example shows how to machine an involute spline with module = 1 (deviating from DIN 3960).

Program sequence

- Tool call: Internal gear cutter
- Start turning mode
- Reset the coordinate system with Cycle **801**
- Move to safe position
- Define Cycle **285**
- Call Cycle **287**
- Reset the coordinate system with Cycle **801**

0 BEGIN PGM 7 MM	
1 BLK FORM CYLINDER Z D90 L35 DIST+0 DI58	
2 TOOL CALL "SKIVING"	; Call the tool
3 FUNCTION MODE TURN	; Activate turning mode
4 CYCL DEF 801 RESET ROTARY COORDINATE SYSTEM	
5 M145	; Cancel a potentially still active M144
6 FUNCTION TURNDATA SPIN VCONST:OFF S50	; Constant surface speed OFF
7 M140 MB MAX	; Retract the tool
8 L A+0 R0 FMAX	; Set the rotary axis to 0
9 L X+0 Y+0 R0 FMAX	; Pre-position the tool at the workpiece center
10 L Z+50 R0 FMAX	; Pre-position the tool in the spindle axis
11 CYCL DEF 285 DEFINE GEAR~	
Q551=+0 ;STARTING POINT IN Z ~	
Q552=-11 ;END POINT IN Z ~	
Q540=+1 ;MODULE ~	
Q541=+90 ;NUMBER OF TEETH ~	
Q542=+90 ;OUTSIDE DIAMETER ~	
Q563=+1 ;TOOTH HEIGHT ~	
Q543=+0.05 ;TROUGH-TIP CLEARANCE ~	
Q544=-10 ;ANGLE OF INCLINATION	
12 CYCL DEF 287 GEAR SKIVING ~	
Q240=+5 ;NUMBER OF CUTS ~	
Q584=+1 ;NO. OF FIRST CUT ~	
Q585=+5 ;NO. OF LAST CUT ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q260=+50 ;CLEARANCE HEIGHT ~	
Q545=+20 ;TOOL LEAD ANGLE ~	
Q546=+0 ;CHANGE ROTATION DIR. ~	
Q547=+0 ;ANG. OFFSET, SPINDLE ~	
Q550=+1 ;MACHINING SIDE ~	
Q533=+1 ;PREFERRED DIRECTION ~	

Q530=+2 ;INCLINED MACHINING ~	
Q253=+2222 ;F PRE-POSITIONING ~	
Q586=+0,4 ;FIRST INFEED ~	
Q587=+0,1 ;LAST INFEED ~	
Q588=+0,4 ;FIRST FEED RATE ~	
Q589=+0,25 ;LAST FEED RATE ~	
Q580=+0,2 ;FEED-RATE ADAPTION	
13 CYCL CALL M303	; Call the cycle, spindle ON
14 FUNCTION MODE MILL	; Activate milling mode
15 M140 MB MAX	; Retract the tool in the tool axis
16 L A+0 C+0 R0 FMAX	; Reset the rotation
17 M30	; End of program
18 END PGM 7 MM	

15

Coordinate Transformation

15.1 Reference systems

15.1.1 Overview

A control requires unambiguous coordinates in order to move an axis to a defined position correctly. For coordinates to be unambiguous, they not only require the values but also a reference system in which these values are valid.

The control differentiates between the following reference systems:

Abbreviation	Meaning	Further information
M-CS	Machine coordinate system machine coordinate system	Page 935
B-CS	Basic coordinate system basic coordinate system	Page 938
W-CS	Workpiece coordinate system workpiece coordinate system	Page 940
WPL-CS	Working plane coordinate system working plane coordinate system	Page 942
I-CS	Input coordinate system input coordinate system	Page 944
T-CS	Tool coordinate system tool coordinate system	Page 946

The control uses different reference systems for different purposes. For example, this makes it possible to always exchange tools at the exact same position while maintaining the possibility of adapting an NC program to the workpiece position.

The reference systems build upon each other. The machine coordinate system **M-CS** is the fundamental reference system. The position and orientation of the following reference systems are determined by transformations of the M-CS.

Definition

Transformations
Translatory transformations each enable a shift along a number line. Rotatory transformations enable a rotation around a point.

15.1.2 Basics of coordinate systems

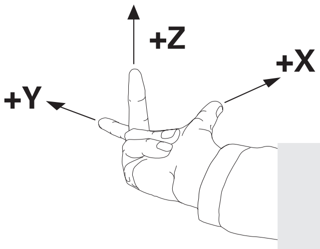
Types of coordinate systems

For coordinates to be unambiguous they must define one point in all axes of the coordinate system:

Axes	Function
One	In a one-dimensional coordinate system, one coordinate defines one point on a number line. Example: on a machine tool, a linear encoder represents a number line.
Two	In a two-dimensional coordinate system, two coordinates define one point in a plane.
Three	In a three-dimensional coordinate system, three coordinates define one point in space.

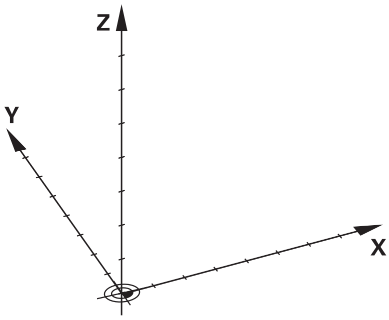
If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

Using the right-hand rule you can recreate a three-dimensional Cartesian coordinate system. The fingertips point in the positive directions of the three axes.



Origin of the coordinate system

Unambiguous coordinates require a defined reference point to which the values refer, starting from zero. This point is the coordinate origin, which lies at the intersection of the axes for all three-dimensional Cartesian coordinate systems of the control. The coordinate origin has the coordinates **X+0**, **Y+0**, and **Z+0**.



15.1.3 Machine coordinate system M-CS

Application

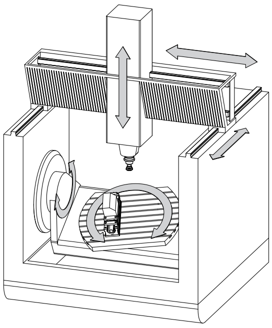
In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

Description of function

Properties of M-CS machine coordinate system

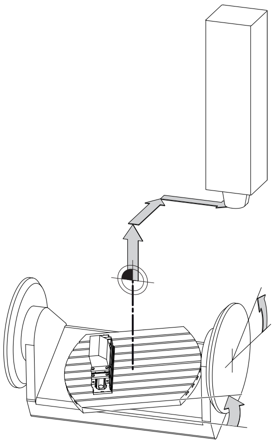
The machine coordinate system **M-CS** corresponds to the kinematics description and therefore to the actual mechanical design of the machine tool. The physical axes of a machine tool are not necessarily always exactly perpendicular to each other, and therefore do not represent a Cartesian coordinate system. The **M-CS** thus consists of multiple one-dimensional coordinate systems that correspond to the axes of the machine.

The machine manufacturer defines the position and orientation of the one-dimensional coordinate systems in the kinematics description.



The machine datum is the coordinate origin of the **M-CS**. The machine manufacturer defines the machine datum in the machine configuration.

The values in the machine configuration define the zero positions of the position encoders and the corresponding machine axes. The machine datum does not necessarily have to be located in the theoretical intersection of the physical axes. It can also be located outside of the traverse range.



Position of the machine datum in the machine

Transformations in the machine coordinate system M-CS

The following transformations can be defined in the **M-CS** machine coordinate system:

- Axis-specific shifts in the **OFFS** columns of the preset table
  Further information: "Preset table", Page 1851

  > The machine manufacturer configures the **OFFS** columns of the preset table in accordance with the machine.

- The **Additive offset (M-CS)** function for rotary axes in the **GS** workspace (option 44)
  Further information: "Global Program Settings (GPS, option 44)", Page 1132

> The machine manufacturer can also define further transformations.
> **Further information:** "Note", Page 937

Position display

The following modes of the position display are referenced to the machine coordinate system **M-CS**:

- **Nominal reference position (RFNOML)**
- **Actual reference position (RFACTL)**

The difference between the values for the **RFACTL** and **ACTL.** modes of an axis result from all stated offsets as well as all active transformations in other reference systems.

Programming coordinate entry in machine coordinate system M-CS

With miscellaneous function **M91** you program the coordinates relative to the machine datum.

Further information: "Traversing in the machine coordinate system M-CS with M91", Page 1228

Note

The machine manufacturer can define the following further transformations in the machine coordinate system **M-CS**:

- Additive axis shifts for parallel axes with the **OEM-offset**
- Axis-specific shifts in the **OFFS** columns of the pallet preset table
  Further information: "Pallet preset table", Page 1775

> **NOTICE**
>
> **Danger of collision!**
>
> The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. Since the values of the pallet preset table are neither visible nor editable, there is a risk of collision during any movement!
>
> - Refer to the machine manufacturer's documentation
> - Use pallet presets only in conjunction with pallets

Example

This example illustrates the difference between traverse movements with and without **M91**. The example shows the behavior with a Y axis as oblique axis that is not arranged perpendicularly to the ZX plane.

Traverse movement without M91

```
11 L IY+10
```

You use the Cartesian input coordinate system **I-CS** for programming. The **ACTL.** and **NOML.** modes of the position display show only a movement of the Y axis in the **I-CS**.

The control uses the defined values to determine the required traverse paths of the machine axes. Since the machine axes are not arranged perpendicularly to each other, the control moves the axes **Y** and **Z**.

Since the machine coordinate system **M-CS** is a projection of the machine axes, the **RFACTL** and **RFNOML** modes of the position display show movements of the Y axis and Z axis in the **M-CS**.

Traverse movement with M91

```
11 L IY+10 M91
```

The control moves the machine axis **Y** by 10 mm. The **RFACTL** and **RFNOML** modes of the position display show only a movement of the Y axis in the **M-CS**.

In contrast to the **M-CS**, the **I-CS** is a Cartesian coordinate system; the axes of the two reference systems do not coincide. The **ACTL.** and **NOML.** modes of the position display show movements of the Y axis and Z axis in the **I-CS**.

15.1.4 Basic coordinate system B-CS

Application

In the basic coordinate system **B-CS** you define the position and orientation of the workpiece. You determine these values by using a 3D touch probe, for example. The control saves the values in the preset table.

Description of function

Properties of the basic coordinate system B-CS

The basic coordinate system **B-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the end of the kinematics description.

The machine manufacturer defines the coordinate origin and orientation of the **B-CS**.

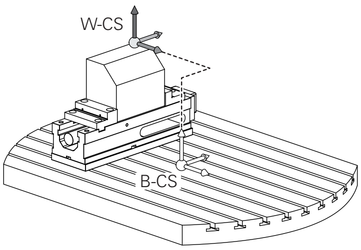
Transformations in the basic coordinate system B-CS

The following columns of the preset table have an effect in the basic coordinate system **B-CS**:

- **X**
- **Y**
- **Z**
- **SPA**
- **SPB**
- **SPC**

You determine the position and orientation of the workpiece coordinate system **W-CS** by using a 3D touch probe, for example. The control saves the determined values as basic transformations in the **B-CS** in the preset table.

Further information: "Preset management", Page 949



The machine manufacturer configures the **BASE TRANSFORM.** columns of the preset table in accordance with the machine.

The machine manufacturer can also define further transformations.

Further information: "Note", Page 939

Note

The machine manufacturer can define additional basic transformations and store them in the pallet preset table.

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. Since the values of the pallet preset table are neither visible nor editable, there is a risk of collision during any movement!

- Refer to the machine manufacturer's documentation
- Use pallet presets only in conjunction with pallets

15.1.5 Workpiece coordinate system W-CS

Application

In the workpiece coordinate system **W-CS** you define the position and orientation of the working plane. You do this by programming transformations and tilting the working plane.

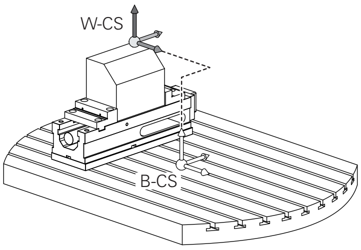
Description of function

Properties of the workpiece coordinate system W-CS

The workpiece coordinate system **W-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the active workpiece preset from the preset table.

Both the position and orientation of the **W-CS** are defined by basic transformations in the preset table.

Further information: "Preset management", Page 949



Transformations in the workpiece coordinate system (W-CS)

HEIDENHAIN recommends using the following transformations in the workpiece coordinate system **W-CS**:

- **TRANS DATUM** function before tilting the working plane

  Further information: "Datum shift with TRANS DATUM", Page 972
- Function **TRANS MIRROR** or Cycle **8 MIRRORING** before tilting the working plane with spatial angles

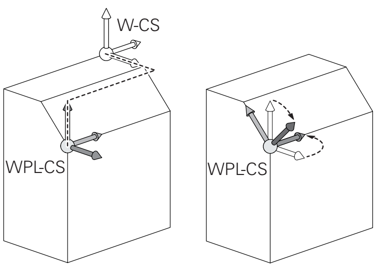
  Further information: "Mirroring with TRANS MIRROR", Page 973

  Further information: "Cycle 8 MIRRORING", Page 959
- **PLANE** functions for tilting the working plane (option 8)

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 979

> The control also offers Cycle **19 WORKING PLANE** for tilting the working plane.

With these transformations, the position and orientation of the working plane coordinate system **WPL-CS** are changed.



NOTICE

Danger of collision!

The control reacts differently to the various types of transformations as well as their programmed sequence. Unexpected movements or collisions can occur if the functions are not suitable.

- Program only the recommended transformations in the respective reference system
- Use tilting functions with spatial angles instead of with axis angles
- Use the Simulation mode to test the NC program

In the machine parameter **planeOrientation** (no. 201202) the machine manufacturer defines whether the control interprets input values of Cycle **19 WORKING PLANE** as spatial angles or as axis angles.

The type of tilting function has the following effects on the result:

- If you tilt using spatial angles (**PLANE** functions except for **PLANE AXIAL** or Cycle **19**), previously programmed transformations will change the position of the workpiece datum and the orientation of the rotary axes:
  - Shifting with the **TRANS DATUM** function will change the position of the workpiece datum.
  - Mirroring changes the orientation of the rotary axes. The entire NC program, including the spatial angles, will be mirrored.
- If you tilt using axis angles (**PLANE AXIAL** or Cycle **19**), a previously programmed mirroring has no effect on the orientation of the rotary axes. You use these functions for direct positioning of the machine axes.

Additional transformations with Global Program Settings (GPS, option 44)

In the **GS** workspace (option 44) you can define additional transformations in the workpiece coordinate system **W-CS**:

- **Additive basic rotat. (W-CS)**

  The effects of this function are added to a basic rotation or a 3D basic rotation from the preset table or the pallet preset table. This function is the first transformation that is possible in the **W-CS**.
- **Shift (W-CS)**

  This function is in effect in addition to a datum shift defined in the NC program with the **TRANS DATUM** function and before the working plane is tilted.
- **Mirroring (W-CS)**

  The function is effective in addition to a mirror image (function **TRANS MIRROR** or Cycle **8 MIRRORING**) defined in the NC program and before tilting the working plane.
- **Shift (mW-CS)**

  This function is in effect in the modified workpiece coordinate system. This function is in effect after the **Shift (W-CS)** and **Mirroring (W-CS)** functions and before the working plane is tilted.

Further information: "Globale Programmeinstellungen GPS", Page

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.

  Further information: "Input coordinate system I-CS", Page 944
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.

  Further information: "Working plane coordinate system WPL-CS", Page 942
- The result of transformations built upon each other depends on the programming sequence.

15.1.6 Working plane coordinate system WPL-CS

Application

In the working plane coordinate system **WPL-CS** you define the position and orientation of the input coordinate system **I-CS** and therefore the reference for the coordinate system in the NC program. You do this by programming transformations after having tilted the working plane.

Further information: "Input coordinate system I-CS", Page 944

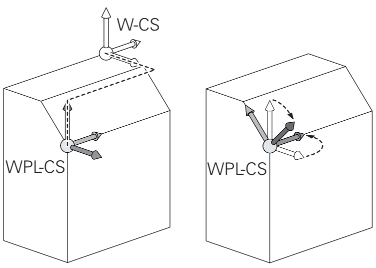
Description of function

Properties of the working plane coordinate system WPL-CS

The working plane coordinate system **WPL-CS** is a three-dimensional Cartesian coordinate system. You use transformations in the workpiece coordinate system **W-CS** to define the coordinate origin of the **WPL-CS**.

Further information: "Workpiece coordinate system W-CS", Page 940

If no transformations are defined in the **W-CS**, then the position and orientation of the **W-CS** and **WPL-CS** are identical.



Transformations in the working plane coordinate system WPL-CS

HEIDENHAIN recommends using the following transformations in the working plane coordinate system **WPL-CS**:

- **TRANS DATUM** function

  Further information: "Datum shift with TRANS DATUM", Page 972
- **TRANS MIRROR** or Cycle **8 MIRRORING** function

  Further information: "Mirroring with TRANS MIRROR", Page 973

  Further information: "Cycle 8 MIRRORING", Page 959
- **TRANS ROTATION** function or Cycle **10 ROTATION**

  Further information: "Rotations with TRANS ROTATION", Page 975

  Further information: "Cycle 10 ROTATION ", Page 961
- **TRANS SCALE** function or Cycle **11 SCALING FACTOR**

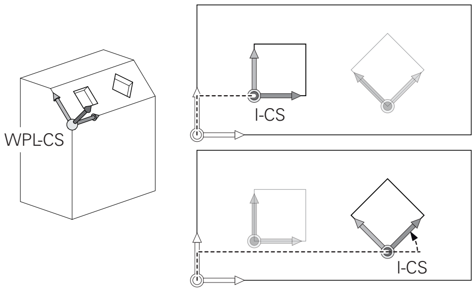
  Further information: "Scaling with TRANS SCALE", Page 977

  Further information: "Cycle 11 SCALING FACTOR ", Page 962
- Cycle **26 AXIS-SPECIFIC SCALING**

  Further information: "Cycle 26 AXIS-SPECIFIC SCALING ", Page 963
- **PLANE RELATIV** function (option 8)

  Further information: "PLANE RELATIV", Page 1004

With these transformations you modify the position and orientation of the input coordinate system **I-CS**.



NOTICE

Danger of collision!

The control reacts differently to the various types of transformations as well as their programmed sequence. Unexpected movements or collisions can occur if the functions are not suitable.

- Program only the recommended transformations in the respective reference system
- Use tilting functions with spatial angles instead of with axis angles
- Use the Simulation mode to test the NC program

Additional transformations with Global Program Settings (GPS, option 44)

The **Rotation (WPL-CS)** transformation in the **GS** workspace is in effect in addition to a rotation in the NC program.

Further information: "Globale Programmeinstellungen GPS", Page

Additional transformations with mill-turning (option 50)

The following additional transformations are available with the mill-turning software option:

- Precession angle with the following cycles:
  - Cycle **800 ADJUST XZ SYSTEM**
  - Cycle **801 RESET ROTARY COORDINATE SYSTEM**
  - Cycle **880 GEAR HOBBING**
- OEM transformations defined by machine manufacturers for special turning kinematics

> Machine manufacturers can also define an OEM transformation and a precession angle without software option 50.
>
> An OEM transformation takes effect before the precession angle.
>
> If an OEM transformation or a precession angle is defined, the control shows the values on the **POS** tab of the **Status** workspace. These transformations are also in effect in milling mode!
>
> **Further information:** "POS tab", Page 157

Additional transformation with Gear Cutting (option 157)

You can use the following cycles to define a precession angle:

- Cycle **286 GEAR HOBBING**
- Cycle **287 GEAR SKIVING**

> Machine manufacturers can also define a precession angle without Gear Cutting (software option 157)

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.

  Further information: "Input coordinate system I-CS", Page 944
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.
- The result of transformations built upon each other depends on the programming sequence.
- As a **PLANE** function (option 8), **PLANE RELATIV** has an effect in the workpiece coordinate system **W-CS** and orients the working plane coordinate system **WPL-CS**. The values of additive tilting always relate to the current **WPL-CS**.

15.1.7 Input coordinate system I-CS

Application

The programmed values in the NC program refer to the input coordinate system **I-CS**. You use positioning blocks to program the position of the tool.

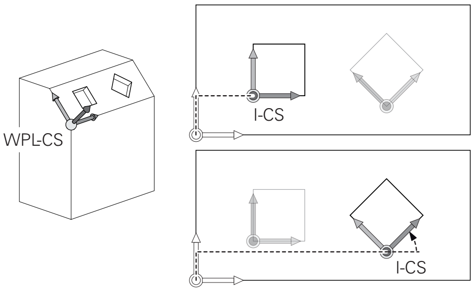
Description of function

Properties of the input coordinate system I-CS

The input coordinate system **I-CS** is a three-dimensional Cartesian coordinate system. You use transformations in the working plane coordinate system **WPL-CS** to define the coordinate origin of the **I-CS**.

Further information: "Working plane coordinate system WPL-CS", Page 942

If no transformations are defined in the **WPL-CS**, then the position and orientation of the **WPL-CS** and **I-CS** are identical.



Positioning blocks in the input coordinate system I-CS

In the input coordinate system **I-CS** you use positioning blocks to define the position of the tool. The position of the tool defines the position of the tool coordinate system **T-CS**.

Further information: "Tool coordinate system T-CS", Page 946

You can define the following positioning blocks:

- Paraxial positioning blocks
- Path functions with Cartesian or polar coordinates
- Straight lines **LN** with Cartesian coordinates and surface normal vectors (option 9)
- Cycles

11 X+48 R+	; Paraxial positioning block

11 L X+48 Y+102 Z-1.5 R0	; Path function **L**

11 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 R0	; Straight line **LN** with Cartesian coordinates and surface normal vector

Position display

The following modes of the position display are referenced to the input coordinate system **I-CS**:

- **Nominal pos. (NOML)**
- **Actual pos. (ACT)**

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.

  Further information: "Working plane coordinate system WPL-CS", Page 942

15.1.8 Tool coordinate system T-CS

Application

In the tool coordinate system **T-CS** the control implements tool compensations and tool inclinations.

Description of function

Properties of the tool coordinate system T-CS

The tool coordinate system **T-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the tool tip TIP.

You make entries in the tool management to define the tool tip relative to the tool carrier reference point. The machine manufacturer usually defines the tool carrier reference point on the spindle tip.

Further information: "Presets in the machine", Page 187

You define the tool tip with the following columns of the tool management relative to the tool carrier reference point:

- **L**
- **DL**
- **ZL** (option 50, option 156)
- **XL** (option 50, option 156)
- **YL** (option 50, option 156)
- **DZL** (option 50, option 156)
- **DXL** (option 50, option 156)
- **DYL** (option 50, option 156)
- **LO** (option 156)
- **DLO** (option 156)

Further information: "Tool carrier reference point", Page 245

You use positioning blocks in the input coordinate system **I-CS** to define the position of the tool and therefore the position of the **T-CS**.

Further information: "Input coordinate system I-CS", Page 944

You can use miscellaneous functions to also program in other reference systems, such as **M91** for the machine coordinate system **M-CS**.

Further information: "Traversing in the machine coordinate system M-CS with M91", Page 1228

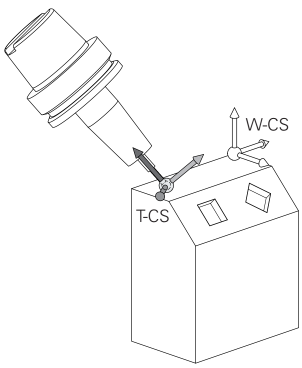
The orientation of the **T-CS** in most cases is identical to that of the **I-CS**.

If the following functions are active, the orientation of the **T-CS** depends on the tool angle of inclination:

- Miscellaneous function **M128** (option 9)

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1245
- **PLANE RELATIV** function (option 9)

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027



Use the miscellaneous function **M128** to define the tool angle of inclination in the machine coordinate system **M-CS** using axis angles. The effects of the tool angle of inclination depend on the machine kinematics:

Further information: "Notes", Page 1249

11 L X+10 Y+45 A+10 C+5 R0 M128	; Straight line with miscellaneous function **M128** and axis angles

You can also define a tool angle of inclination in the working plane coordinate system **WPL-CS**, for example with **FUNCTION TCPM** or a straight line **LN**.

11 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS	; **FUNCTION TCPM** with spatial angles
12 L A+0 B+45 C+0 R0 F2500	

11 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 TX-0.08076201 TY-0.34090025 TZ0.93600126 R0 M128	; Straight line **LN** with surface normal vector and tool orientation

Transformations in the tool coordinate system T-CS

The following tool compensations have an effect in the tool coordinate system **T-CS**:

- Compensation values from the tool management

  Further information: "Tool compensation for tool length and radius", Page 1036
- Compensation values from the tool call

  Further information: "Tool compensation for tool length and radius", Page 1036
- Values of the compensation tables ***.tco**

  Further information: "Tool compensation with compensation tables", Page 1044
- Values of **FUNCTION TURNDATA CORR T-CS** (option 50)

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1047
- 3D tool compensation with surface normal vectors (option 9)

  Further information: "3D tool compensation (option 9)", Page 1049
- 3D tool radius compensation depending on the tool's contact angle using compensation-value tables (option 92)

  Further information: "3D radius compensation depending on the tool contact angle (option 92)", Page 1062

Position display

The display of the virtual tool axis **VT** refers to the tool coordinate system **T-CS**.

The control shows the values of **VT** in the **GS workspace** (option 44) and on the **GS** tab of the **Status** workspace.

Further information: "Global Program Settings (GPS, option 44)", Page 1132

The HR 520 and HR 550 FS handwheels show the values of **VT** in the display.

Further information: "Contents of an electronic handwheel display", Page 1884

15.2 Preset management

Application

The preset management allows setting and activating single presets. The presets to be saved may include e. g. the position and the misalignment of a workpiece in the preset table. The active row of the preset table serves as the workpiece preset in the NC program and as the origin of workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 187

Use the preset management in the following cases:

- To tilt the working plane of a machine with table or head rotation axes (option 8)
- To work on a machine with a head change system
- To machine several workpieces that are clamped down at different misaligned positions
- If REF-based datum tables were used on previous control models

Related topics

- Contents of preset table, write protection

  Further information: "Preset table", Page 1851

Description of function

Setting presets

Presets can be set in the following ways:

- Setting axis positions manually

  Further information: "Setting a preset manually", Page 952
- Touch probe cycles in the **Setup** application

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1425
- Touch probe cycles in the NC program

  Further information: "Programmable Touch Probe Cycles", Page 1449

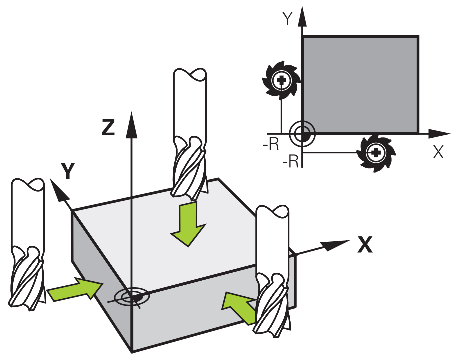
  Further information: "Cycle 247 PRESETTING ", Page 969

If you try to write a value in a write-protected preset table row, the control cancels this process with an error message. Write-protection for this row must be rescinded first.

Further information: "Removing write protection", Page 1854

Setting a preset with milling cutters

If no workpiece touch probe is available, the preset can also be set by using a milling cutter. In this case, the values are not obtained by probing, but by scratching.



When scratching with a milling cutter, the tool is slowly approached to the workpiece edge in the **Manual operation** application while the spindle is rotating.

As soon as the tool produces chips on the workpiece, the preset is manually set in the desired axis.

Further information: "Setting a preset manually", Page 952

Activating presets

NOTICE
Caution: Significant property damage! Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. ▶ Before activating a preset, check whether all columns contain values.

Presets can be activated in the following ways:

- Activating manually in the **Tables** operating mode

  Further information: "Activating a preset manually", Page 953
- Cycle **247 PRESETTING**

  Further information: "Cycle 247 PRESETTING ", Page 969
- **PRESET SELECT** function

  Further information: "Activating the preset with PRESET SELECT", Page 954

When activating a preset, the control resets the following transformations:

- Datum shift with the **TRANS DATUM** function
- Mirror image with the **TRANS MIRROR** function or Cycle **8 MIRRORING**
- Rotation with the **TRANS ROTATION** function or Cycle **10 ROTATION**
- Scaling with the **TRANS SCALE** function or Cycle **11 SCALING FACTOR**
- Axis-specific scaling with Cycle **26 AXIS-SPECIFIC SCALING**

Tilting the working plane by using **PLANE** functions or Cycle **19 WORKING PLANE** will not be reset by the control.

Basic rotation and 3D basic rotation

The **SPA**, **SPB** and **SPC** columns define a spatial angle for orienting the workpiece coordinate system **W-CS**. This spatial angle defines the basic rotation or 3D basic rotation of the preset.

Further information: "Workpiece coordinate system W-CS", Page 940

When a rotation around the tool axis is defined, the preset contains a basic rotation, e. g. **SPC** for tool axis **Z**. If one of the remaining columns is defined, the preset contains a 3D basic rotation. If the workpiece preset contains a basic rotation or 3D basic rotation, the control takes these values into account when executing an NC program.

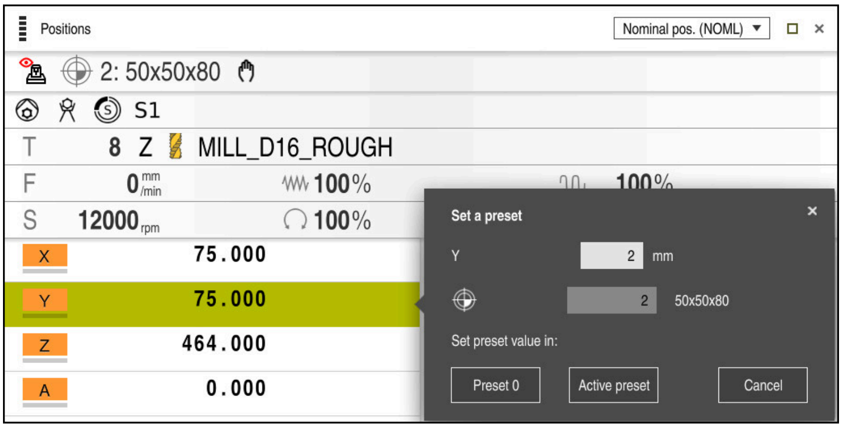
The **3D ROT** button (option 8) allows defining that the control takes a basic rotation or 3D basic rotation into account in the **Manual operation** application as well.

Further information: "3-D rotation window (option 8)", Page 1022

When a basic rotation or 3D basic rotation is active, the control displays a symbol in the **Positions** workspace.

Further information: "Active functions", Page 144

15.2.1 Setting a preset manually



Set a preset window in the **Positions** workspace

When setting the preset manually, the values can be written either in row 0 of the preset table or in the active row.

To set a preset manually in an axis:

- Select the **Manual operation** application in the **Manual** operating mode
- Open the **Positions** workspace
- Traverse the tool to the desired position, e. g. scratching
- Select the row of the desired axis
- > The control opens the **Set a preset** window.
- Enter the value of the current axis position, relating to the new preset, e. g. **0**
- > The control activates the **Preset 0** and **Active preset** buttons as options.

- Select an option, e. g. **Active preset**
- > The control saves the value in the selected preset table row and closes the **Set a preset** window.
- > The control updates the values in the **Positions** workspace.

- The **Set the preset** button in the function bar opens the **Set a preset** window for the row marked in green.
- When selecting **Preset 0**, the control automatically activates row 0 of the preset table as the workpiece preset.

15.2.2 Activating a preset manually

> ***NOTICE***
>
> **Caution: Significant property damage!**
>
> Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.
>
> - Before activating a preset, check whether all columns contain values.

To activate a preset manually:

- Select the **Tables** operating mode
- Select the **Presets** application
- Select the desired row

Activate the preset

- Select **Activate the preset**
- > The control activates the preset.
- > The control shows the number of the active preset in the **Positions** workspace and in the status overview.

Further information: "Description of function", Page 141

Further information: "Status overview on the control bar", Page 147

Notes

- The machine manufacturer uses the optional **initial** machine parameter (no. 105603) to define a default value for each column of a new row.
- In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (option 8). If the rotary axes are positioned so that they differ from the definition in the **3-D rotation** window, the control cancels this process with an error message by default.

  Further information: "3-D rotation window (option 8)", Page 1022
  In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.
- When scratching a workpiece with the radius of a milling cutter, the radius value must be taken into account in the preset.
- Even if the current preset contains a basic rotation or a 3D basic rotation, the **PLANE RESET** function will position the rotary axes at 0° in the **MDI** application.

  Further information: "Application MDI", Page 1759
- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.

  Further information: "Pallet preset table", Page 1775

15.3 NC functions for preset management

15.3.1 Overview

The control provides the following functions for modifying a preset directly in the NC program after it has been defined in the preset table:

- Activate the preset
- Copy the preset
- Correct the preset

15.3.2 Activating the preset with PRESET SELECT

Application

The **PRESET SELECT** function allows you to use a preset defined in the preset table and activate it as a new preset.

Requirement

- The preset table contains values
  Further information: "Preset management", Page 949
- Workpiece preset has been defined
  Further information: "Setting a preset manually", Page 952

Description of function

To activate the preset, use the preset number or the entry in the **Doc** column. If the entry in the **Doc** column is not unique, the control will activate the preset with the smallest preset number.

The **KEEP TRANS** syntax element allows defining that the control retains the transformations below:

- the **TRANS DATUM** function
- Cycle **8 MIRRORING** and the **TRANS MIRROR** function
- Cycle **10 ROTATION** and the **TRANS ROTATION** function
- Cycle **11 SCALING FACTOR** and the **TRANS SCALE** function
- Cycle **26 AXIS-SPECIFIC SCALING**

Input

11 PRESET SELECT #3 KEEP TRANS WP	; Activate row 3 of the table as the workpiece preset and maintain transformations

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET SELECT	Syntax initiator for activating a preset
#, **" "** or **QS**	Select the row of the preset table Fixed or variable number or name The row can be selected from a selection menu. For names the control only displays the rows in the preset table where the **Doc** column is defined.
KEEP TRANS	Retain simple transformations Optional syntax element
WP or **PAL**	Activate the preset for the workpiece or pallet Optional syntax element

Note

If you program **PRESET SELECT** without optional parameters, the behavior is the same as with Cycle **247 SET PRESET**.

Further information: "Cycle 247 PRESETTING ", Page 969

15.3.3 Copying the preset with PRESET COPY

Application

The function **PRESET COPY** allows you to copy a preset defined in the preset table and activate the preset copied.

Requirement

- The preset table contains values

  Further information: "Preset management", Page 949
- Workpiece preset has been defined

  Further information: "Setting a preset manually", Page 952

Description of function

To select the preset to be copied, use the preset number or the entry in the **Doc** column. If the entry in the **Doc** column is not unique, the control will select the preset with the smallest preset number.

Input

11 PRESET COPY #1 TO #3 SELECT TARGET KEEP TRANS	; Copy row 1 of the preset table to row 3, activate row 3 as the workpiece preset and maintain transformations

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET COPY	Syntax initiator for copying and activating a workpiece preset
#, **" "** or **QS**	Select the row of the preset table to be copied Fixed or variable number or name The row can be selected from a selection menu. For names the control only displays the rows in the preset table where the **Doc** column is defined.
TO #, **" "** or **QS**	Select the new row of the preset table Fixed or variable number or name The row can be selected from a selection menu. For names the control only displays the rows in the preset table where the **Doc** column is defined.
SELECT TARGET	Activate the copied row of the preset table as the workpiece preset Optional syntax element
KEEP TRANS	Optional syntax element

15.3.4 Correcting the preset with PRESET CORR

Application

The function **PRESET CORR** allows you to correct the active preset.

Requirement

- The preset table contains values

  Further information: "Preset management", Page 949
- Workpiece preset has been defined

  Further information: "Setting a preset manually", Page 952

Description of function

If both the basic rotation and a translation are corrected in an NC block, the control will first correct the translation and then the basic rotation.

The compensation values are given with respect to the active coordinate system. When correcting the OFFS values, the values are referenced to the machine coordinate system **M-CS**.

Further information: "Reference systems", Page 934

Input

11 PRESET CORR X+10 SPC+45	; Correct the workpiece preset in **X** by +10 mm and in **SPC** by +45°

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET CORR	Syntax initiator for correcting the workpiece preset
X, **Y**, **Z**	Compensation values in the principal axes Optional syntax element
SPA, **SPB**, **SPC**	Compensation values for the spatial angle Optional syntax element
X_OFFS, **Y_OFFS**, **Z_OFFS**, **A_OFFS**, **B_OFFS**, **C_OFFS**, **U_OFFS**, **V_OFFS**, **W_OFFS**	Compensation value for the offsets, referenced to the machine datum Optional syntax element

15.4 Datum table

Application

A datum table saves positions on the workpiece. To use a datum table, you must activate it. Within an NC program, the datums can be called, e. g. to execute machining processes on several workpieces at the same position. The active row of the preset table serves as the workpiece preset in the NC program.

Related topics

- Contents and preparing a datum table

  Further information: "Datum table", Page 1858
- Editing a datum table during a program run

  Further information: "Compensation during program run", Page 1795
- Preset table

  Further information: "Preset table", Page 1851

Description of function

The datums from a datum table are referenced to the current workpiece preset. The coordinate values from datum tables are only effective as absolute coordinate values.

Datum tables can be used in the following situations:

- Frequent use of the same datum shift
- Recurring machining sequences on different workpieces
- Recurring machining sequences at different positions on the workpiece

Activating the datum table manually

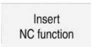
A datum table can be activated manually for the **Program Run** operating mode.

In the **Program Run** operating mode, the **Program settings** window contains the **Tables** area. In this area, a datum table and both compensation tables can be selected in one selection window for running the program.

When activating a table, the control will highlight this table with the status **M**.

15.4.1 Activating the datum table in the NC program

To activate a datum table in the NC program:



- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **SEL TABLE**
- > The control opens the action bar.

- Select **Selection**
- > A file selection window opens.
- Select datum table

- Select **Select**

If the datum table is not stored in the same directory as the NC program, the complete path name must be defined. The **Program settings** window allows defining whether the control creates absolute or relative paths.

Further information: "Settings in the Program workspace", Page 197

> If you enter the datum table name manually, please note the following:
> - If the datum table is stored in the same directory as the NC program, enter the file name only.
> - If the datum table is not stored in the same directory as the NC program, enter the complete path.

Definition

File format	Definition
.d	Datum table

15.5 Coordinate transformation cycles

15.5.1 Fundamentals

Once a contour has been programmed, the control can execute it on the workpiece at various locations and in different sizes by using cycles for coordinate transformation.

Effectiveness of coordinate transformations

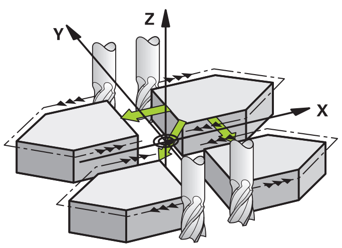
Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called separately. It remains in effect until it is changed or canceled.

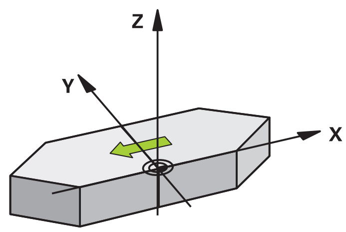
Reset coordinate transformation:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM NC block (these M functions depend on the machine parameters)
- Select a new NC program

15.5.2 Cycle 8 MIRRORING

Application





The control can machine the mirror image of a contour in the working plane.

Mirroring becomes effective as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed; this does not apply to SL cycles
- If you mirror two axes, the machining direction remains the same.

The result of the mirroring depends on the location of the datum:

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also "jumps" to another location.

Reset

Program Cycle **8 MIRRORING** again with **NO ENT**.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

i	For working in a tilted system with Cycle **8**, the following procedure is recommended: ■ **First** program the tilting movement and **then** call Cycle **8 MIRRORING**!

Cycle parameters

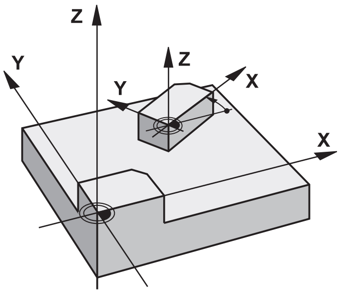
Help graphic	Parameter
	Mirror image axis? Enter the axis to be mirrored. You can mirror all axes—including rotary axes—with the exception of the spindle axis and its associated secondary axis. You can enter up to three NC axes. Input: **X**, **Y**, **Z**, **U**, **V**, **W**, **A**, **B**, **C**

Example

```
11 CYCL DEF 8.0 MIRRORING
12 CYCL DEF 8.1 X Y Z
```

15.5.3 Cycle 10 ROTATION

Application



Within an NC program, the control can rotate the coordinate system in the working plane about the active datum.

The ROTATION cycle becomes effective as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. The active angle of rotation is shown in the additional status display.

Reference axis for the rotation angle:

- X/Y plane: X axis
- Y/Z plane: Y axis
- Z/X plane: Z axis

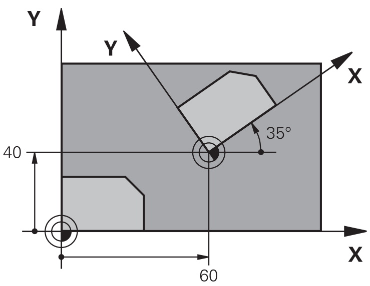
Reset

Program Cycle **10 ROTATION** again and specify a rotation angle of 0°.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **10** cancels an active radius compensation. If necessary, reprogram the radius compensation.
- After defining Cycle **10**, move both axes of the working plane to activate the rotation for all axes.

Cycle parameters

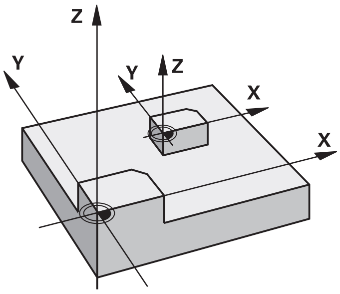
Help graphic	Parameter
	**Rotation angle?** Enter the angle of rotation in degrees (°). Enter the value as an incremental or absolute value. Input: **-360.000...+360.000**

Example

```
11 CYCL DEF 10.0 ROTATION
12 CYCL DEF 10.1  ROT+35
```

15.5.4 Cycle 11 SCALING FACTOR

Application



The control can increase or reduce the size of contours within an NC program. This enables you to program shrinkage and oversize allowances.

The scaling factor becomes effective as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

- all three coordinate axes at the same time
- dimensions in cycles

Requirement

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Enlargement: SCL greater than 1 (up to 99.999 999)

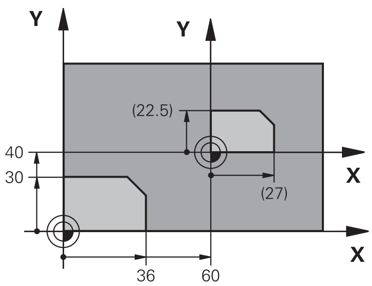
Reduction: SCL less than 1 (down to 0.000 001)

> This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Reset

Program Cycle **11 SCALING FACTOR** again and specify a scaling factor of 1.

Cycle parameters

<table>
<tr><th>Help graphic</th><th>Parameter</th></tr>
<tr><td>


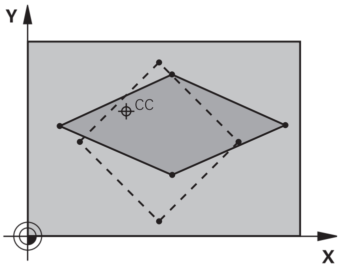
</td><td>Factor?
Enter the scaling factor SCL. The control multiplies the coordinates and radii by the SCL factor (as described under "Effect" above).
Input: 0.000001...99.999999</td></tr>
</table>

Example

```
11 CYCL DEF 11.0 SCALING FACTOR
12 CYCL DEF 11.1 SCL 0.75
```

15.5.5 Cycle 26 AXIS-SPECIFIC SCALING

Application



Use Cycle **26** to account for shrinkage and allowance factors for each axis.

The scaling factor becomes effective as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. The active scaling factor is shown in the additional status display.

Reset

Program Cycle **11 SCALING FACTOR** again and enter a scaling factor of 1 for the corresponding axis.

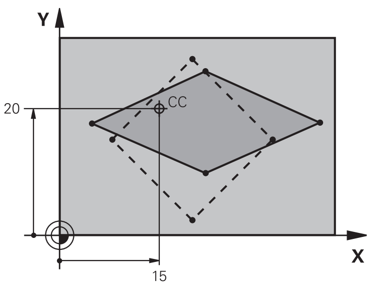
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The contour is enlarged or reduced relative to the center, and not necessarily (as in Cycle **11 SCALING FACTOR**) relative to the active datum.

Notes on programming

- Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.
- You can program each coordinate axis with its own axis-specific scaling factor.
- In addition, you can enter the coordinates of a center for all scaling factors.

Cycle parameters

Help graphic	Parameter
	**Axis and factor?** Select the coordinate axis/axes via the action bar. Enter the factor(s) for axis-specific enlargement or reduction. Input: **0.000001...99.999999**
	Centerpoint coord. of extension? Center of the axis-specific enlargement or reduction. Input: **-999999999...+999999999**

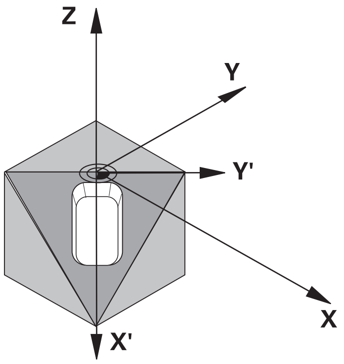
Example

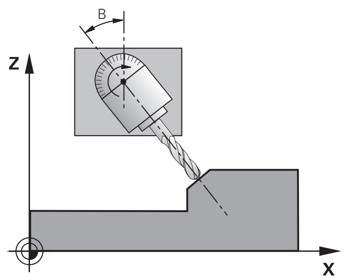
```
11 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
12 CYCL DEF 26.1 X1.4 Y0.6 CCX+15 CCY+20
```

15.5.6 Cycle 19 WORKING PLANE (option 8)

Application

	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.





Use Cycle **19** to define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the rotary axes directly.
- Describe the position of the working plane using up to three rotations (spatial angles) of the **machine-based** coordinate system.

  The required spatial angles can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt. With two spatial angles, every tool position in space can be defined exactly.

	Note that the position of the tilted coordinate system, and therefore also all movements in the tilted system, are dependent on your description of the tilted plane.

If you program the position of the working plane via spatial angles, the control will calculate the required angle positions of the tilted axes automatically and will store these in the **Q120** (A axis) to **Q122** (C axis) parameters. If two solutions are possible, the control will choose the shorter path from the current position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The control first rotates the A axis, then the B axis, and finally the C axis.

Cycle **19** becomes effective as soon as it has been defined in the NC program. As soon as you move an axis in the tilted system, the compensation for this specific axis will be activated. You must move all axes to activate compensation for all axes.

If you set the **Program Run** switch in the **3D ROT** menu (**Manual operation** operating mode / **Manual operation**) application to active, the angular value entered in this menu is overwritten by Cycle **19 WORKING PLANE**.

Notes

- This cycle can be executed in the **FUNCTION MODE MILL** machining mode.
- In combination with a radial facing slide kinematics model, this cycle can also be used in the **FUNCTION MODE TURN** machining mode.
- The working plane is always tilted around the active datum.
- If you use the Cycle **19** while **M120** is active, the control automatically cancels the radius compensation, which also cancels the **M120** function.

Notes on programming

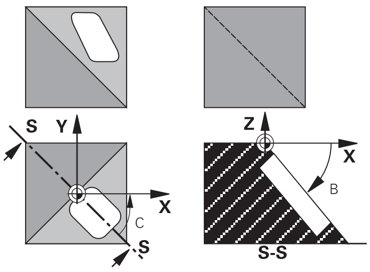
- Write the program as if the machining process was to be executed in a non-tilted plane.
- If you call the cycle again for other angles, you do not need to reset the machining parameters.

> Because nonprogrammed rotary axis values are interpreted as unchanged, you should always define all three spatial angles, even if one or more angles are at zero.

Notes about machine parameters

- The machine manufacturer specifies whether the programmed angles are interpreted by the control as coordinates of the rotary axes (axis angles) or as angular components of a tilted plane (spatial angles).
- In the machine parameter **CfgDisplayCoordSys** (no. 127501) the machine manufacturer defines the coordinate system in which the status display shows an active datum shift.

Cycle parameters

Help graphic	Parameter
	**Rotary axis and angle?** Enter the axis of rotation together with the associated tilt angles. Program the rotary axes A, B and C using the action bar. Input: **-360.000...+360.000**

If the control automatically positions the rotary axes, you can enter the following parameters:

Help graphic	Parameter
	Feed rate? F= Traverse speed of the rotary axis during automatic positioning Input: 0...300000
	Set-up clearance? The control positions the tilting head in such a way that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece. This value has an incremental effect. Input: **0...999999999**

Reset

To reset the tilt angles, redefine Cycle **19 WORKING PLANE**. Enter an angular value of 0° for all rotary axes. Then, redefine Cycle **19 WORKING PLANE**. Confirm the dialog prompt by pressing the **NO ENT** key. This disables the function.

Positioning the axes of rotation

Refer to your machine manual.

The machine manufacturer determines whether Cycle **19** positions the axes of rotation automatically or whether they need to be positioned manually in the NC program.

Manual positioning of rotary axes

If Cycle **19** does not position the rotary axes automatically, you need to position them in a separate L block following the cycle definition.

If you use axis angles, you can define the axis values right in the L block. For using spatial angles, program the Q parameters **Q120** (A axis value), **Q121** (B axis value) and **Q122** (C axis value) according to Cycle **19**.

For manual positioning, always use the rotary axis positions stored in Q parameters **Q120** to **Q122**.

Avoid the use of functions such as **M94** (modulo rotary axes) in order to prevent discrepancies between actual and nominal positions of the rotary axes for multiple calls.

Example

11 L Z+100 R0 FMAX	
12 L X+25 Y+10 R0 FMAX	
*** - ...**	; Define the spatial angles for calculating the compensation
13 CYCL DEF 19.0 WORKING PLANE	
14 CYCL DEF 19.1 A+0 B+45 C+0	
15 L A+Q120 C+Q122 R0 F1000	; Position the rotary axes by using values calculated by Cycle 19
16 L Z+80 R0 FMAX	; Activate compensation for the spindle axis
17 L X-8.5 Y-10 R0 FMAX	; Activate compensation for the working plane

Automatic positioning of rotary axes

If the rotary axes are positioned automatically in Cycle **19**:

- The control can position only closed-loop axes.
- To position the tilted axes, you must enter a feed rate and a set-up clearance, in addition to the tilting angles, when defining the cycle
- Use only preset tools (the full tool length must have been defined)
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting.
- The control performs tilting at the last programmed feed rate (the maximum feed rate depends on the complexity of the swivel head geometry or tilting table)

Example

11 L Z+100 R0 FMAX	
12 L X+25 Y+10 R0 FMAX	
* - ...	; Angle for calculating the compensation; define the feed rate and clearance
13 CYCL DEF 19.0 WORKING PLANE	
14 CYCL DEF 19.1 A+0 B+45 C+0 F5000 ABST50	
15 L Z+80 R0 FMAX	; Activate compensation for the spindle axis
16 L X-8.5 Y-10 R0 FMAX	; Activate compensation for the working plane

Position display in a tilted system

On activation of Cycle **19**, the displayed positions (**NOML** and **ACTL**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. This means that the position displayed immediately after cycle definition might not be the same as the coordinates of the last programmed position before Cycle **19**.

Monitoring of the working space

The control monitors only those axes in the tilted coordinate system that are moved. Where applicable, the control displays an error message.

Positioning in a tilted coordinate system

With miscellaneous function **M130**, you can move the tool, while the coordinate system tilted, to positions that reference the non-tilted coordinate system.

With a tilted working plane, it is also possible to position the axes using straight-line blocks that reference the machine coordinate system (NC blocks with **M91** or **M92**). Constraints:

- Positioning is without length compensation.
- Positioning is done without length compensation.
- Tool radius compensation is not allowed.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is tilted about the active datum. You can program a datum shift before activating Cycle **19**. In this case, you are shifting the machine-based coordinate system.

If you program a datum shift after the activation of Cycle **19** , you are shifting the tilted coordinate system.

Important: When resetting the cycles, reverse the sequence used for defining them:

1 Activate datum shift
2 Activate **Tilt working plane**
3 Activate rotation

...

Workpiece machining

...

1 Reset the rotation
2 Reset **Tilt working plane**
3 Reset the datum shift

Procedure for working with Cycle 19 WORKING PLANE

Proceed as follows:

- Create the NC program
- Clamp the workpiece
- Set any presets
- Start the NC program

Creating the NC program:

- Call the defined tool
- Retract in the spindle axis
- Position the axes of rotation
- Activate a datum shift if required
- Define Cycle **19 WORKING PLANE**
- Position all principal axes (X, Y, Z) in order to activate the compensation
- Define Cycle **19** with different angles, if necessary
- Reset Cycle **19** by programming 0° for all rotary axes
- Redefine Cycle **19** in order to deactivate the working plane
- Reset datum shift if required.
- Position the tilt axes to the 0° position if required.

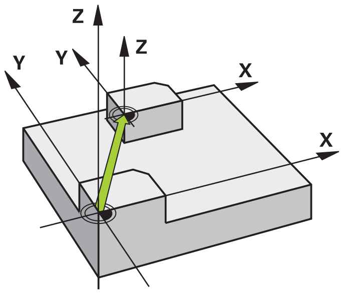
You can define the preset in the following ways:

- Manually by touch-off
- Controlled with a HEIDENHAIN 3-D touch probe
- Automatically with a HEIDENHAIN 3-D touch probe

"Touch Probe Cycles: Automatic Preset Measurement"
"Workpiece presetting with a touch probe"

15.5.7 Cycle 247 PRESETTING

Application



Use Cycle **247 PRESETTING** to activate a preset defined in the preset table as the new preset.

After cycle definition, all coordinate input and datum shifts (absolute or incremental) reference the new preset.

Status display

In **Program Run** the control shows the active preset number behind the preset symbol in the **Positions** workspace.

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- When activating a preset from the preset table, the control resets the datum shift, mirroring, rotation, scaling factor, and axis-specific scaling factor.
- If you activate preset number 0 (line 0), then you activate the preset that you last set in the **Manual operation** operating mode.
- Cycle **247** is also in effect in the simulation.

Cycle parameters

Help graphic	Parameter
	Number for preset? Enter the number of the desired preset from the preset table. Alternatively, you can use the button with the preset symbol in the action bar to directly select the desired preset from the preset table. Input: **0...65535**

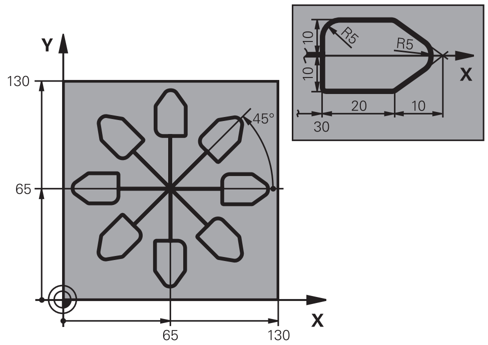
Example

```
11 CYCL DEF 247 PRESETTING ~
    Q339=+4                    ;PRESET NUMBER
```

15.5.8 Example: coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- Machining within a subprogram



0 BEGIN PGM C220 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+130 Y+130 Z+0	
3 TOOL CALL 1 Z S4500	; Tool call
4 L Z+100 R0 FMAX M3	; Retract the tool
5 TRANS DATUM AXIS X+65 Y+65	; Shift datum to center
6 CALL LBL 1	; Call milling operation
7 LBL 10	; Set label for program-section repeat
8 CYCL DEF 10.0 ROTATION	
9 CYCL DEF 10.1 IROT+45	
10 CALL LBL 1	; Call milling operation
11 CALL LBL 10 REP6	; Jump back to LBL 10; repeat six times
12 CYCL DEF 10.0 ROTATION	
13 CYCL DEF 10.1 ROT+0	
14 TRANS DATUM RESET	; Reset datum shift
15 L Z+250 R0 FMAX	; Retract the tool
16 M30	; End program
17 LBL 1	; Subprogram 1
18 L X+0 Y+0 R0 FMAX	; Define milling operation
19 L Z+2 R0 FMAX	
20 L Z-5 R0 F200	
21 L X+30 RL	
22 L IY+10	
23 RND R5	
24 L IX+20	
25 L IX+10 IY-10	
26 RND R5	
27 L IX-10 IY-10	
28 L IX-10 IY-10	

```
29 L IX-20
30 L IY+10
31 L  X+0  Y+0 R0 F5000
32 L  Z+20 R0 FMAX
33 LBL 0
34 END PGM C220 MM
```

15.6 NC functions for coordinate transformation

15.6.1 Overview

The control provides the following **TRANS** functions:

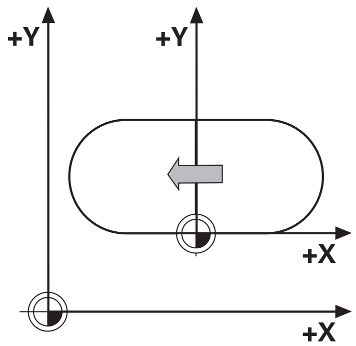
Syntax	Function	Further information
TRANS DATUM	Shift the workpiece datum	Page 972
TRANS MIRROR	Mirror an axis	Page 973
TRANS ROTATION	Rotation about the tool axis	Page 975
TRANS SCALE	Scale contours and positions	Page 977

Define the functions in the sequence in which they are listed in the table and reset them in reverse order. The sequence of programming will have an impact on the result.

For example, if you first shift the workpiece datum and then mirror the contour and then reverse the sequence, the contour will be mirrored at the original workpiece datum.

All **TRANS** functions reference the workpiece datum. The workpiece datum is the origin of the input coordinate system (**I-CS**).

Further information: "Input coordinate system I-CS", Page 944



Related topics

- Coordinate transformation cycles

  Further information: "Coordinate transformation cycles", Page 958
- **PLANE** functions (option 8)

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 979
- Reference systems

  Further information: "Reference systems", Page 934

15.6.2 Datum shift with TRANS DATUM

Application

The **TRANS DATUM** function allows you to shift the workpiece datum by either entering fixed or variable coordinates or by specifying a table row in the datum table.

Use the **TRANS DATUM RESET** function to reset the datum shift.

Related topics

- Contents of the datum table
  Further information: "Datum table", Page 1858
- Activating the datum table
  Further information: "Activating the datum table in the NC program", Page 958
- Machine presets
  Further information: "Presets in the machine", Page 187

Description of function

TRANS DATUM AXIS

You can define a datum shift by entering values in the respective axis with the **TRANS DATUM AXIS** function. You can define up to nine coordinates in one NC block, and incremental entries are possible.

The control displays the result of the datum shift in the **Positions** workspace.

Further information: "Positions workspace", Page 141

TRANS DATUM TABLE

You can use the **TRANS DATUM TABLE** function to define a datum shift by selecting a row from a datum table.

Optionally, you can set the path to a datum table. If you do not define a path, the control will use the datum table that has been activated with **SEL TABLE**.

Further information: "Activating the datum table in the NC program", Page 958

The control displays the datum shift and the path to the datum table on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 160

TRANS DATUM RESET

Use the **TRANS DATUM RESET** function to cancel a datum shift. How you previously defined the datum is irrelevant.

Input

11 TRANS DATUM AXIS X+10 Y+25 Z+42	; Shift the workpiece datum in the **X**, **Y** and **Z** axes

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS DATUM	Start of syntax for a datum shift
AXIS, **TABLE** or **RESET**	Datum shift with coordinate input, with a datum table or reset of the datum shift
X, **Y**, **Z**, **A**, **B**, **C**, **U**, **V** or **W**	Possible axes for coordinate input Fixed or variable number Only if **AXIS** has been selected
TABLINE	Row in the datum table Fixed or variable number Only if **TABLE** has been selected
" " or **QS**	Path to the datum table Fixed or variable name Optional syntax element Only if **TABLE** has been selected

Notes

- The **TRANS DATUM** function replaces Cycle **7 DATUM SHIFT**. If you import an NC program from a previous control model, the control converts Cycle **7** into the **TRANS DATUM** NC function.
- Absolute values reference the workpiece preset. Incremental values reference the workpiece datum.

  Further information: "Presets in the machine", Page 187
- In machine parameter **transDatumCoordSys** (no. 127501), the machine manufacturer defines the reference system referred to by the values in the position display.

  Further information: "Reference systems", Page 934

15.6.3 Mirroring with TRANS MIRROR

Application

Use the **TRANS MIRROR** function to mirror contours or positions about one or more axes.

The **TRANS MIRROR RESET** function allows you to reset the mirroring.

Related topics

- Cycle **8 MIRRORING**

  Further information: "Cycle 8 MIRRORING", Page 959
- Additive mirroring within the global program settings GPS (option 44)

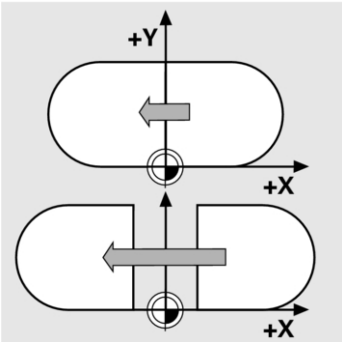
  Further information: "Function Mirroring (W-CS)", Page 1138

Description of function

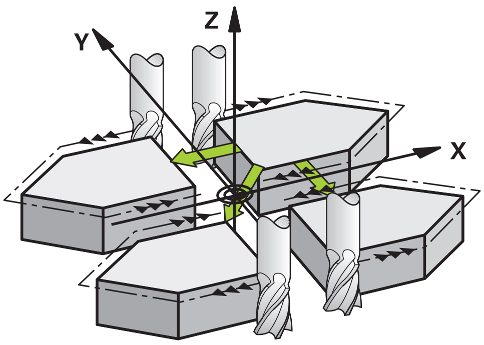
Mirroring is a modal function that is effective as soon as it has been defined in the NC program.

The control mirrors contours or positions about the active workpiece datum. If the datum is outside the contour, the control will also mirror the distance to the datum.

Further information: "Presets in the machine", Page 187



If you mirror only one axis, the machining direction of the tool is reversed The rotational direction defined in a cycle will remain unchanged, e.g. if defined within one of the OCM cycles (option 167).

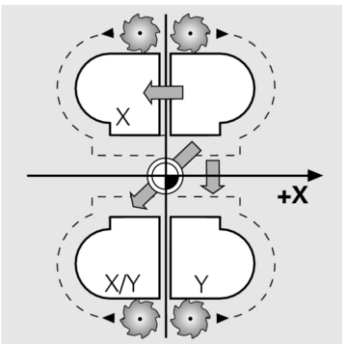


Depending on the selected **AXIS** axis values, the control will mirror the following working planes:

- **X**: The control mirrors the **YZ** working plane
- **Y**: The control mirrors the **ZX** working plane
- **Z**: The control mirrors the **XY** working plane

Further information: "Designation of the axes on milling machines", Page 186

You can select up to three axis values.



If mirroring is active, the control displays it on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 160

Input

11 TRANS MIRROR AXIS X	; Mirror the machining operations about the X axis

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS MIRROR	Start of syntax for mirroring
AXIS or **RESET**	Enter mirroring of axis values or reset mirroring
X, **Y** or **Z**	Axis values to be mirrored Only if **AXIS** has been selected

Note

This function can only be used in the **FUNCTION MODE MILL** machining mode.

Further information: "Switching the operating mode with FUNCTION MODE", Page 210

Notes on using these functions in conjunction with tilting functions

> ***NOTICE***
>
> **Danger of collision!**
>
> The control reacts differently to the various types of transformations as well as their programmed sequence. Unexpected movements or collisions can occur if the functions are not suitable.
>
> - Program only the recommended transformations in the respective reference system
> - Use tilting functions with spatial angles instead of with axis angles
> - Use the Simulation mode to test the NC program

The type of tilting function has the following effects on the result:

- If you tilt using spatial angles (**PLANE** functions except for **PLANE AXIAL** or Cycle **19**), previously programmed transformations will change the position of the workpiece datum and the orientation of the rotary axes:
  - Shifting with the **TRANS DATUM** function will change the position of the workpiece datum.
  - Mirroring changes the orientation of the rotary axes. The entire NC program, including the spatial angles, will be mirrored.
- If you tilt using axis angles (**PLANE AXIAL** or Cycle **19**), a previously programmed mirroring has no effect on the orientation of the rotary axes. You use these functions for direct positioning of the machine axes.

Further information: "Workpiece coordinate system W-CS", Page 940

15.6.4 Rotations with TRANS ROTATION

Application

With the **TRANS ROTATION** function, you can rotate contours or positions around a rotation angle.

The **TRANS DATUM RESET** function allows you to reset the rotation.

Related topics

- Cycle **10 ROTATION**
  Further information: "Cycle 10 ROTATION ", Page 961
- Additive rotation within the global program settings GPS (option 44)

Description of function

Rotation is a modal function that is effective as soon as it has been defined in the NC program.

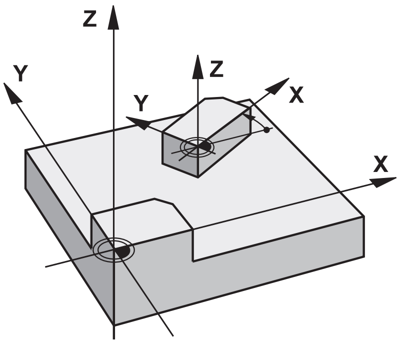
The control rotates machining in the working plane about the active workpiece datum.

Further information: "Presets in the machine", Page 187

The control rotates the input coordinate system (**I-CS**) as follows:

- Based on the angle reference axis, i.e. the main axis
- About the tool axis

Further information: "Designation of the axes on milling machines", Page 186



A rotation can be programmed as follows:

- Absolute, relative to the positive main axis
- Incremental, relative to the last active rotation

If rotation is active, the control displays it on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 160

Input

11 TRANS ROTATION ROT+90	; Rotate machining by 90°

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS ROTATION	Start of syntax for a rotation
ROT or **RESET**	Enter an absolute or incremental angle of rotation or reset rotation Fixed or variable number

Note

This function can only be used in the **FUNCTION MODE MILL** machining mode.

Further information: "Switching the operating mode with FUNCTION MODE", Page 210

15.6.5 Scaling with TRANS SCALE

Application

With the **TRANS SCALE** function, you can scale contours or positions and thus enlarge or reduce these uniformly. This enables you to program shrinkage and oversize allowances, for example.

Use the **TRANS SCALE RESET** function to reset the scaling.

Related topics

- Cycle **11 SCALING FACTOR**
  Further information: "Cycle 11 SCALING FACTOR ", Page 962

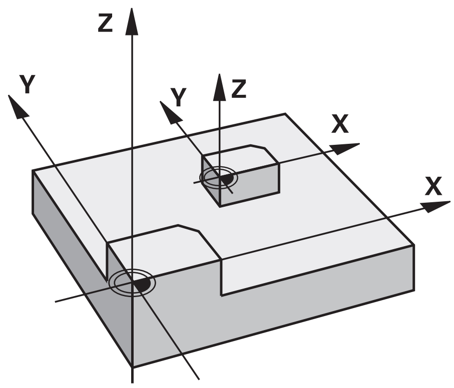
Description of function

Scaling is a modal function that is effective as soon as it has been defined in the NC program.

Depending on the position of the workpiece datum, scaling is carried out as follows:

- Workpiece datum at the center of the contour:
  The contour is scaled uniformly in all directions.
- Workpiece datum at the bottom left of the contour:
  The contour is scaled in the positive X and Y axis directions.
- Workpiece datum at the top right of the contour:
  The contour is scaled in the negative X and Y axis directions.

Further information: "Presets in the machine", Page 187



If you enter a scaling factor **SCL** less than 1, the contour will be reduced in size. If you enter a scaling factor **SCL** greater than 1, the contour will be enlarged.

When scaling, the control takes the coordinate input and dimensions from all cycles into account.

If scaling is active, the control displays it on the **TRANS** tab of the **Status** workspace.

Further information: "TRANS tab", Page 160

Input

11 TRANS SCALE SCL1.5	; Enlarge the contour by the factor 1.5

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS SCALE	Start of syntax for scaling
SCL or **RESET**	Enter the scaling factor or reset scaling Fixed or variable number

Notes

- This function can only be used in the **FUNCTION MODE MILL** machining mode.
  Further information: "Switching the operating mode with FUNCTION MODE", Page 210
- If you want to reduce the size of a contour with inside radii, make sure to select an appropriate tool. Otherwise, residual material might remain.

15.7 Tilting the working plane (option 8)

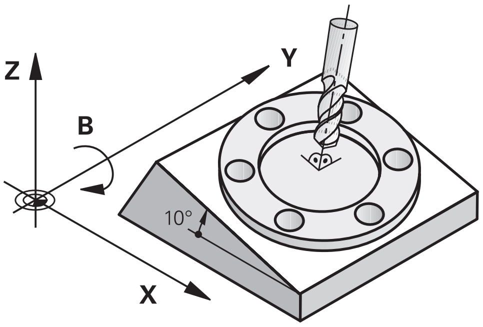
15.7.1 Fundamentals

Machines with rotary axes allow machining e. g. several workpiece sides after one clamping process by tilting the working plane. The tilting functions also allow aligning a workpiece clamped at an incorrect angle.

The working plane can be tilted only when tool axis **Z** is active.

The control functions for tilting the working plane are coordinate transformations. The working plane is always perpendicular to the direction of the tool axis.

Further information: "Working plane coordinate system WPL-CS", Page 942



There are three functions available for tilting the working plane:

- Manual tilting with the **3-D rotation** window in the **Manual operation** application
  Further information: "3-D rotation window (option 8)", Page 1022
- Tilting under program control with the **PLANE** functions in the NC program
  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 979
- Tilting under program control with Cycle **19 WORKING PLANE**
  Further information: "Cycle 19 WORKING PLANE (option 8)", Page 964

Notes concerning different machine kinematics

When no transformations are active and the working plane is not tilted, the linear machine axes move in parallel with the basic coordinate system **B-CS**. In this process, machines behave almost identically, regardless of the kinematics.

Further information: "Basic coordinate system B-CS", Page 938

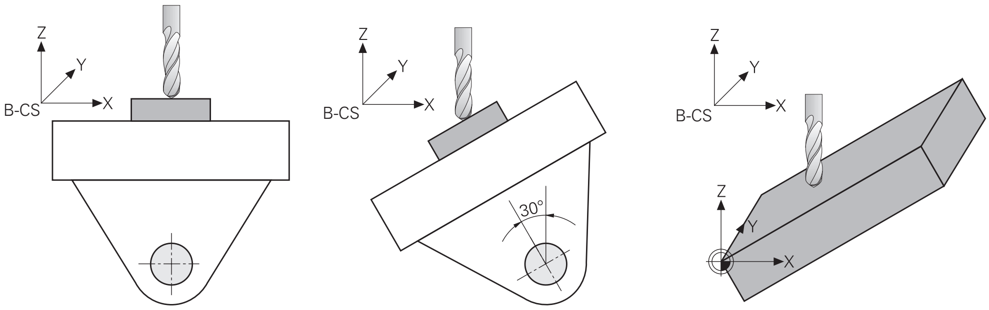
When tilting the working plane, the control moves the machine axes according to the kinematics.

Please observe the aspects below regarding the machine kinematics:

- Machine with table rotary axes

  With this kinematics, the table rotary axes execute the tilting movement and the position of the workpiece in the work envelope changes. The linear machine axes move in the tilted working plane coordinate system **WPL-CS** just as they do in the non-tilted **B-CS**.

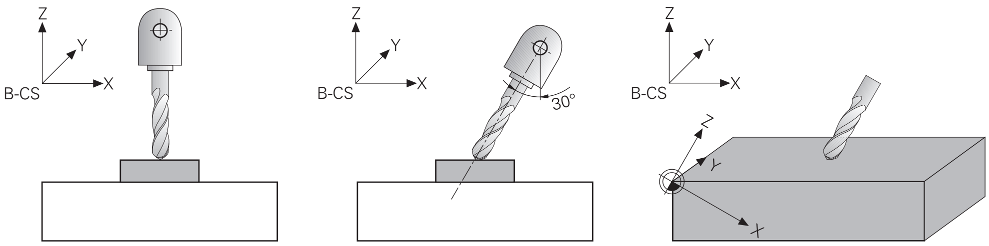
  Further information: "Working plane coordinate system WPL-CS", Page 942



- Machine with head rotary axes

  With this kinematics, the head rotary axes execute the tilting movement and the position of the workpiece in the work envelope remains the same. In the tilted **WPL-CS**, at least two linear machine axes no longer move in parallel with the non-tilted **B-CS**, depending on the rotary angle.

  Further information: "Working plane coordinate system WPL-CS", Page 942



15.7.2 Tilting the working plane with PLANE functions (option 8)

Fundamentals

Application

Machines with rotary axes allow machining e. g. several workpiece sides after one clamping process by tilting the working plane.

The tilting functions also allow aligning a workpiece clamped at an incorrect angle.

Related topics

- Machining types by number of axes

  Further information: "Types of machining according to number of axes", Page 1211
- Adopt tilted working plane in the **Manual** operating mode with the **3-D rotation** window

  Further information: "3-D rotation window (option 8)", Page 1022

Requirements

- Machine with rotary axes

  3+2 axes machining requires at least two rotary axes. Removable axes as an additional top table are also possible.
- Kinematics description

  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 1 (software option 8)
- Tool with tool axis **Z**

Description of function

Tilting the working plane defines the orientation of the working plane coordinate system **WPL-CS**.

Further information: "Reference systems", Page 934

> The position of the workpiece datum and consequently the orientation of the working plane coordinate system **WPL-CS** can be defined by using the **TRANS DATUM** function before tilting the working plane in the workpiece coordinate system **W-CS**.
>
> A datum shift is always effective in the active **WPL-CS**, meaning after the tilting function if applicable. If the workpiece datum is shifted for the tilting process, an active tilting function may have to be reset.
>
> **Further information:** "Datum shift with TRANS DATUM", Page 972

In practice, workpiece drawings show different specified angles, which is why the control offers different **PLANE** functions with different options for defining angles.

Further information: "Overview of PLANE functions", Page 981

In addition to the geometric definition of the working plane, every **PLANE** function allows specifying how the control positions the rotary axes.

Further information: "Rotary axis positioning", Page 1013

If the geometric definition of the working plane results in no unambiguous tilting position, the desired tilting solution can be selected.

Further information: "Tilting solution", Page 1016

Depending on the defined angles and the machine kinematics, there is a choice whether the control positions the rotary axes or orients the working plane coordinate system **WPL-CS** exclusively.

Further information: "Transformation types", Page 1020

Status display

Positions workspace

As soon as the working plane has tilted, the General status display in the **Positions** workspace contains an icon.

Further information: "Positions workspace", Page 141

> When deactivating or resetting the tilting function correctly, the icon indicating the tilted working plane must disappear.
>
> **Further information:** "PLANE RESET", Page 1008

Status workspace

When the working plane is tilted, the **POS** and **TRANS** tabs in the **Status** workspace contain information about the active orientation of the working plane.

When defining the working plane by using axis angles, the control displays the defined axis values. All alternative geometric definition options display the resulting spatial angles.

Further information: "POS tab", Page 157

Further information: "TRANS tab", Page 160

Overview of PLANE functions

The control provides the following **PLANE** functions:

Syntax element	Function	Further information
SPATIAL	Defines the working plane by means of three spatial angles	Page 984
PROJECTED	Defines the working plane by means of two projection angles and one rotation angle	Page 989
EULER	Defines the working plane by means of three Euler angles	Page 994
VECTOR	Defines the working plane by means of two vectors	Page 997
POINTS	Defines the working plane by means of the coordinates of three points	Page 1000
RELATIV	Defines the working plane by means of a single spatial angle with incremental effect	Page 1004
AXIAL	Defines the working plane by means of a maximum of three absolute or incremental axis angles	Page 1009
RESET	Resets tilting of the working plane	Page 1008

Notes

NOTICE

Danger of collision!

When the machine is switched on, the control tries to restore the switch-off status of the tilted plane. This is prevented under certain conditions. For example, this applies if axis angles are used for tilting while the machine is configured with spatial angles, or if you have changed the kinematics.

- If possible, reset tilting before shutting the system down
- Check the tilted condition when switching the machine back on

NOTICE

Danger of collision!

Cycle **8 MIRRORING** can have different effects in conjunction with the **Tilt working plane** function. The programming sequence, the mirrored axes, and the tilting function used are critical in this regard. There is a risk of collision during the tilting operation and subsequent machining!

- Check the sequence and positions using a graphic simulation
- Carefully test the NC program or program section in the **Program run, single block** operating mode

Examples

1. When Cycle **8 MIRRORING** is programmed before the tilting function without rotary axes:
   - The tilt of the **PLANE** function used (except **PLANE AXIAL**) is mirrored
   - Mirroring takes effect after tilting with **PLANE AXIAL** or Cycle **19**
2. When Cycle **8 MIRRORING** is programmed before the tilting function with a rotary axis:
   - The mirrored rotary axis has no effect on the tilt specified in the **PLANE** function used, because only the movement of the rotary axis is mirrored

NOTICE

Danger of collision!

Rotary axes with Hirth coupling must move out of the coupling to enable tilting. There is a danger of collision while the axis moves out of the coupling and during the tilting operation.

- Make sure to retract the tool before changing the position of the rotary axis

- If you use the **PLANE** function when **M120** is active, the control automatically rescinds the radius compensation, which also rescinds the **M120** function.
- Always use **PLANE RESET** to cancel **PLANE** functions. Entering 0 in all **PLANE** parameters (e.g. all three spatial angles) exclusively resets the angles, but not the function.
- If you restrict the number of tilting axes with the **M138** function, your machine may provide only limited tilting possibilities. The machine manufacturer will decide whether the control takes the angles of deselected axes into account or sets them to 0.
- The control only supports tilting the working plane with spindle axis Z.

Tilting the working plane without rotary axes

Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

The machine manufacturer must take the precise angle into account, e.g. the angle of a mounted angle head in the kinematics description.

You can also orient the programmed working plane perpendicularly to the tool without defining rotary axes, e.g. when adapting the working plane for a mounted angle head.

Use the **PLANE SPATIAL** function and the **STAY** positioning behavior to swivel the working plane to the angle specified by the machine manufacturer.

Example of mounted angle head with permanent tool direction **Y**:

Example

```
11 TOOL CALL 5 Z S4500
12 PLANE SPATIAL SPA+0 SPB-90 SPC+0 STAY
```

The tilt angle must be precisely adapted to the tool angle, otherwise the control will generate an error message.

PLANE SPATIAL

Application

Use the **PLANE SPATIAL** function to define the working plane by three spatial angles.

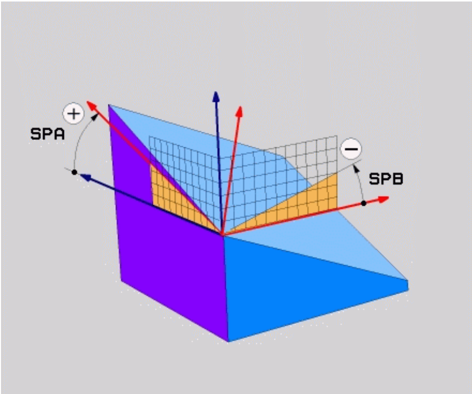
> Spatial angles are the most frequently used definition option for a working plane. The definition is not machine-specific, meaning that it is independent of the rotary axes actually present.

Related topics

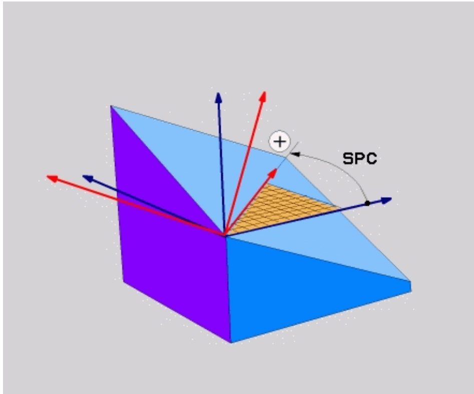
- Defining a single spatial angle with incremental effect
  Further information: "PLANE RELATIV", Page 1004
- Entering the axis angle
  Further information: "PLANE AXIAL", Page 1009

Description of function

Spatial angles define a working plane through three independent rotations in the workpiece coordinate system (**W-CS**), i. e. in the non-tilted working plane.



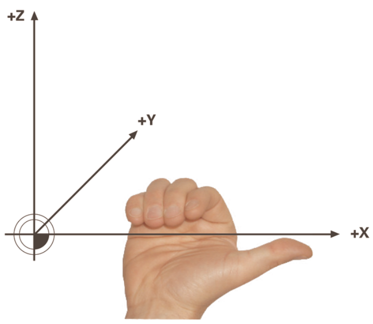
Spatial angles **SPA** and **SPB**



Spatial angle **SPC**

All three angles must be defined even if one or several angles equals 0.

As the spatial angles are programmed independently of the physically existing rotary axes, there is no need to differentiate between the head and the table axes as far as the signs are concerned. Always use the extended right-hand rule.



The thumb of your right hand points in the positive direction of the axis around which the rotation occurs. If you curl your fingers, the curled fingers point in the positive direction of rotation.

Entering the spatial angles as three independent rotations in the workpiece coordinate system **W-CS** in the programming sequence **A-B-C** is a challenge to many users. The challenge in particular is to take two coordinate systems into account simultaneously: the unmodified **W-CS** and the modified working plane coordinate system **WPL-CS**.

This is why the spatial angle can be alternatively defined by imagining three rotations layered on top of one another in the tilting sequence **C-B-A**. This alternative allows considering one coordinate system exclusively, meaning the modified working plane coordinate system **WPL-CS**.

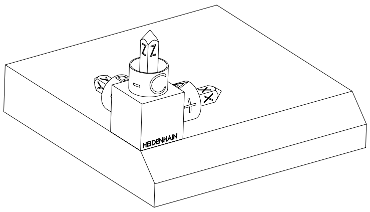
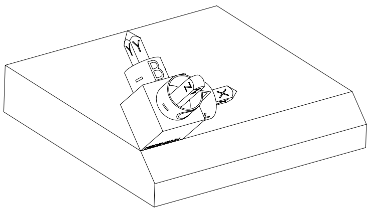
Further information: "Notes", Page 988

> This view equals three **PLANE RELATIV** functions programmed one-by-one, first with **SPC**, then with **SPB** and finally with **SPA**. The spatial angles with incremental effect **SPB** and **SPA** are referenced to the working plane coordinate system **WPL-CS**, i. e. to a tilted working plane.
>
> **Further information:** "PLANE RELATIV", Page 1004

Application example

Example

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis 	Using the defined spatial angle **SPA+45**, the control orients the tilted Z axis of **WPL-CS** to be perpendicular with the chamfer surface. The rotation by the **SPA** angle is around the non-tilted X axis. The orientation of the tilted X axis equals the orientation of the non-tilted X axis. The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced by using four working plane definitions.

If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following spatial angles:

- **SPA+45**, **SPB+0** and **SPC+90** for the second chamfer

  Further information: "Notes", Page 988
- **SPA+45**, **SPB+0** and **SPC+180** for the third chamfer
- **SPA+45**, **SPB+0** and **SPC+270** for the fourth chamfer

The values are referenced to the non-tilted workpiece coordinate system **W-CS**.

Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE SPATIAL	Defines the working plane by means of three spatial angles
SPA	Properties of the workpiece coordinate system **W-CS** Input: **-360.0000000...+360.0000000**
SPB	Properties of the workpiece coordinate system **W-CS** Input: **-360.0000000...+360.0000000**
SPC	Properties of the workpiece coordinate system **W-CS** Input: **-360.0000000...+360.0000000**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1013
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1016 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1020 Optional syntax element

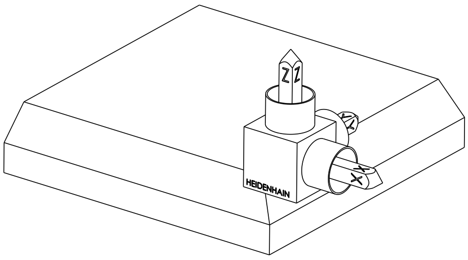
Notes

Comparison of views - Example: chamfer

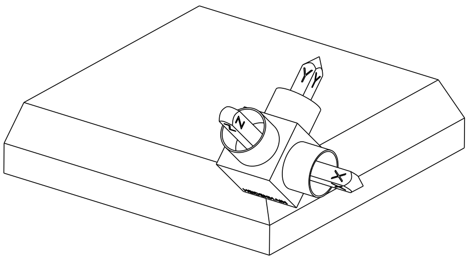
Example

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+90 TURN MB MAX FMAX SYM- TABLE ROT
```

View A-B-C



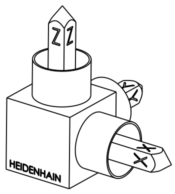
Initial state



SPA+45

Orientation of tool axis **Z**

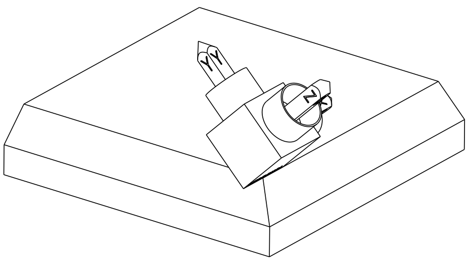
Rotation around the X axis of the non-tilted workpiece coordinate system **W-CS**



SPB+0

Rotation around the Y axis of the non-tilted **W-CS**

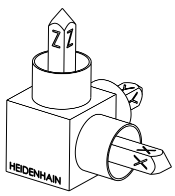
No rotation with value 0



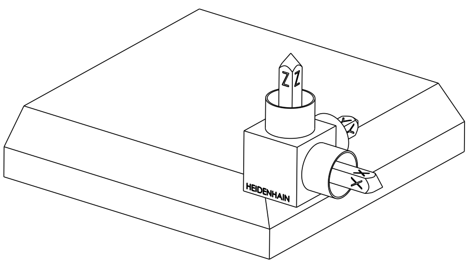
SPC+90

Orientation of main axis **X**

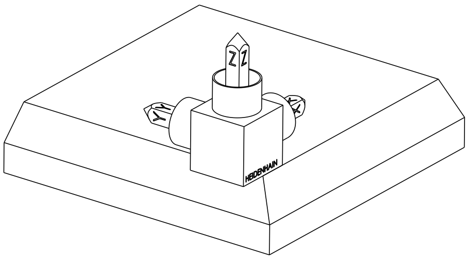
Rotation around the Z axis of the non-tilted **W-CS**



View C-B-A



Initial state



SPC+90

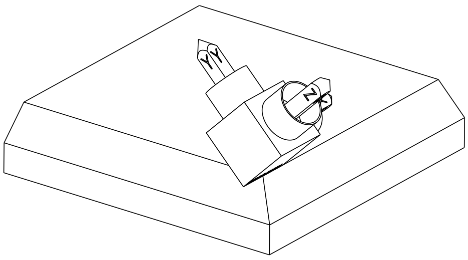
Orientation of main axis **X**

Rotation around the Z axis of the workpiece coordinate system **W-CS**, meaning in the non-tilted working plane

SPB+0

Rotation around the Y axis in the working plane coordinate system **WPL-CS**, meaning in the tilted working plane

No rotation with value 0



SPA+45

Orientation of tool axis **Z**

Rotation around the X axis in **WPL-CS**, meaning in the tilted working plane

Both views have an identical result.

Definition

Abbreviation	Definition
SP e. g. in **SPA**	Spatial

PLANE PROJECTED

Application

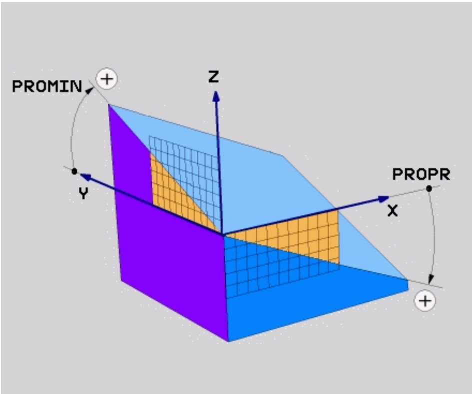
Use the **PLANE PROJECTED** function to define the working plane by two projection angles. Use an additional rotation angle to optionally align the X axis in the tilted working plane.

Description of function

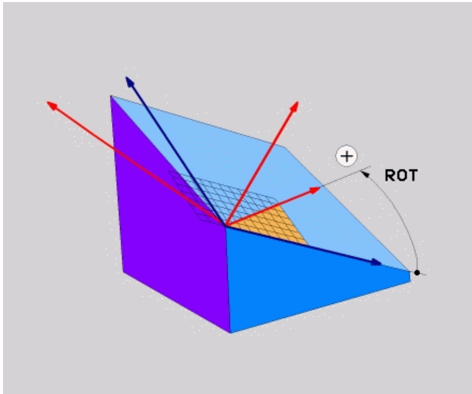
Projection angles define a working plane through two independent angles in the working planes **ZX** and **YZ** of the non-tilted working plane coordinate system **W-CS**.

Further information: "Designation of the axes on milling machines", Page 186

Use an additional rotation angle to optionally align the X axis in the tilted working plane.



Projection angles **PROMIN** and **PROPR**



Rotation angle **ROT**

All three angles must be defined even if one or several angles equals 0.

Entering the projection angles is easy for rectangular workpieces because the workpiece edges are the same as the projection angles.

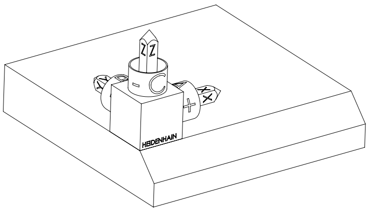
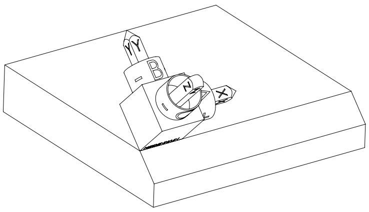
The projection angles of non-rectangular workpieces can be obtained by imagining the working planes **ZX** and **YZ** as transparent panels with angle scales. When viewing the workpiece from the front through the **ZX** plane, the difference between the X axis and the workpiece edge equals the projection angle **PROPR**. Use the same procedure to obtain the projection angle **PROMIN** by viewing the workpiece from the left.

> When using **PLANE PROJECTED** for multi-side or internal machining, the hidden workpiece edges must be used or projected. Imagine the workpiece to be transparent in such cases.
>
> **Further information:** "Notes", Page 993

Application example

Example

```
11 PLANE PROJECTED PROPR+0 PROMIN+45 ROT+0 TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis 	Using the defined projection angle **PROMIN+45**, the control orients the Z axis of **WPL-CS** to be perpendicular with the chamfer surface. The angle from **PROMIN** is effective in the working plane **YZ**. The orientation of the tilted X axis equals the orientation of the non-tilted X axis. The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced by using four working plane definitions.

If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following projection and rotation angles:

- **PROPR+45**, **PROMIN+0** and **ROT+90** for the second chamfer
- **PROPR+0**, **PROMIN-45** and **ROT+180** for the third chamfer
- **PROPR-45**, **PROMIN+0** and **ROT+270** for the fourth chamfer

The values are referenced to the non-tilted workpiece coordinate system **W-CS**.

Remember that the workpiece datum must be shifted before each working plane definition.

Input

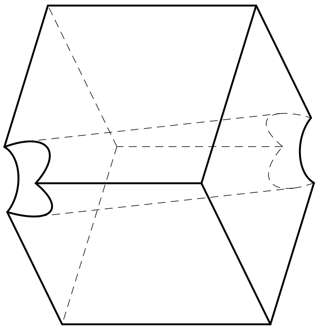
```
11 PLANE PROJECTED PROPR+0 PROMIN+45 ROT+0 TURN MB MAX FMAX SYM- TABLE
   ROT
```

The NC function includes the following syntax elements:

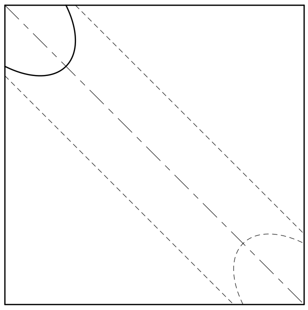
Syntax element	Meaning
PLANE PROJECTED	Syntax initiator for the working plane definition by means of two projection angles and one rotation angle
PROPR	Angle in working plane **ZX**, i. e. around the Y axis of the workpiece coordinate system **W-CS** Input: **-89.999999...+89.9999**
PROMIN	Angle in the working plane **YZ**, i. e. around the X axis of **W-CS** Input: **-89.999999...+89.9999**
ROT	Rotation around the Z axis of the tilted working plane coordinate system **WPL-CS** Input: **-360.0000000...+360.0000000**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1013
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1016 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1020 Optional syntax element

Notes

Procedure in case of hidden workpiece edges, using the example of a diagonal hole



Cube with a diagonal hole

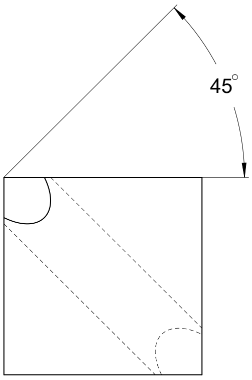


Front view, meaning projection on the **ZX** working plane

Example

```
11 PLANE PROJECTED PROPR-45  PROMIN+45 ROT+0 TURN MB MAX FMAX SYM-
   TABLE ROT
```

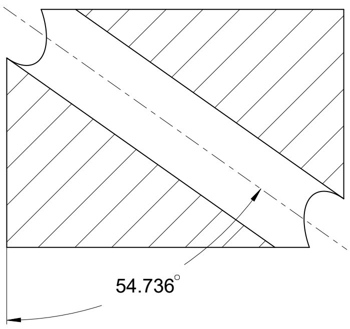
Comparison of projection and spatial angles



When imagining the workpiece to be transparent, the projection angles are easy to find.

Both projection angles are 45°.

> When defining the algebraic sign, ensure that the working plane is perpendicular to the center axis of the hole.



When defining the working plane by using spatial angles, the spatial diagonal must be considered.

The full section along the hole axis shows that the axis does not form an isosceles triangle with the lower and the left workpiece edge. This is why e. g. a spatial angle **SPA+45** produces an incorrect result.

Definition

Abbreviation	Definition
PROPR	Main plane
PROMIN	Minor plane
ROT	Angle of rotation

PLANE EULER

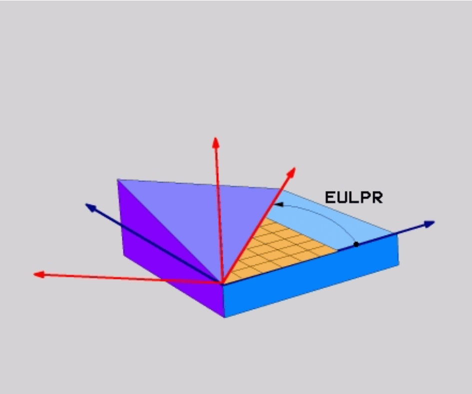
Application

Use the **PLANE EULER** function to define the working plane by three Euler angles.

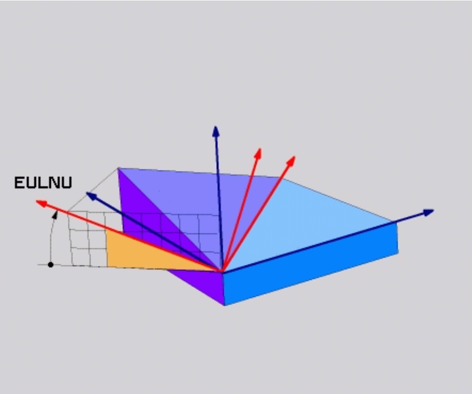
Description of function

Euler angles define a working plane as three rotations layered on top of one another, starting from the non-tilted workpiece coordinate system **W-CS**.

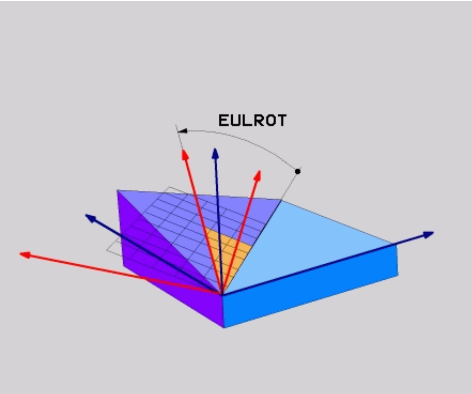
Use the third Euler angle to optionally align the tilted X axis.



Euler angle **EULPR**



Euler angle **EULNU**



Euler angle **EULROT**

All three angles must be defined even if one or several angles equals 0.

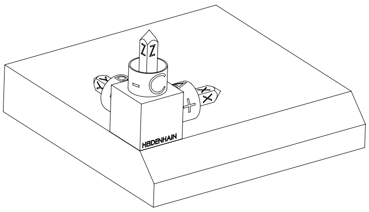
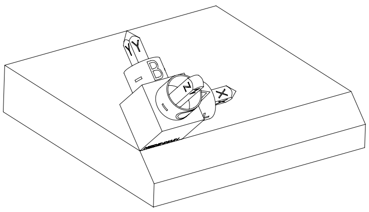
At first, the rotations layered on top of one another happen around the non-tilted Z axis, then around the tilted X axis and finally around the tilted Z axis.

> This view equals three **PLANE RELATIV** functions programmed one-by-one, first with **SPC**, then with **SPA** and finally with **SPC** again.
>
> **Further information:** "PLANE RELATIV", Page 1004
>
> The same result can be achieved by a **PLANE SPATIAL** function with the spatial angles **SPC** and **SPA**, followed by a rotation, e. g. with the **TRANS ROTATION** function.
>
> **Further information:** "PLANE SPATIAL", Page 984
>
> **Further information:** "Rotations with TRANS ROTATION", Page 975

Application example

Example

```
11 PLANE EULER EULPR+0 EULNU45 EULROT0 TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis 	Using the defined Euler angle **EULNU**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by the **EULNU** angle is around the non-tilted X axis. The orientation of the tilted X axis equals the orientation of the non-tilted X axis. The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

> When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced by using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following Euler angles:
>
> - **EULPR+90**, **EULNU45** and **EULROT0** for the second chamfer
> - **EULPR+180**, **EULNU45** and **EULROT0** for the third chamfer
> - **EULPR+270**, **EULNU45** and **EULROT0** for the fourth chamfer
>
> The values are referenced to the non-tilted workpiece coordinate system **W-CS**.
>
> Remember that the workpiece datum must be shifted before each working plane definition.

Input

Example

```
11 PLANE EULER EULPR+0 EULNU45 EULROT0 TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE EULER	Syntax initiator for the working plane definition by means of three Euler angles
EULPR	Rotation around the Z axis of the workpiece coordinate system **W-CS** Input: **-180.000000...+180.000000**
EULNU	Rotation around the X axis of the tilted working plane coordinate system **WPL-CS** Input: **0...180.000000**
EULROT	Rotation around the Z axis of the tilted **WPL-CS** Input: **0...360.000000**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1013
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1016 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1020 Optional syntax element

Definition

Abbreviation	Definition
EULPR	Precession angle
EULNU	Nutation angle
EULROT	Angle of rotation

PLANE VECTOR

Application

Use the **PLANE VECTOR** function to define the working plane by two vectors.

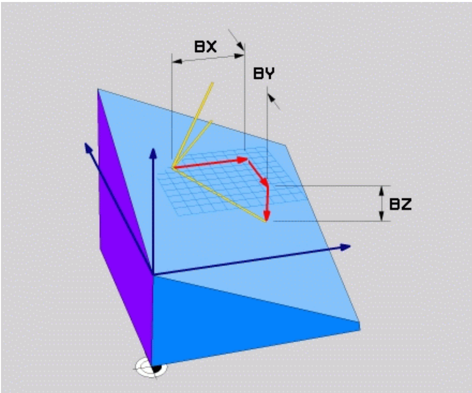
Related topics

- Output formats of NC programs

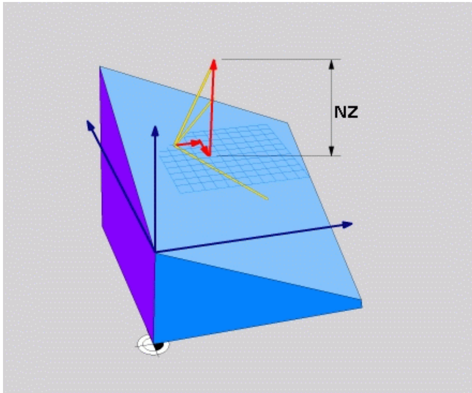
  Further information: "Output formats of NC programs", Page 1209

Description of function

Vectors define a working plane as two independent specifications of direction, starting from the non-tilted workpiece coordinate system **W-CS**.



Base vector with components **BX**, **BY** and **BZ**



NZ component of the normalized vector

All six components must be defined even if one or several components equals 0.

> There is no need to enter a normalized vector. The drawing dimensions or any values which will not alter the ratio between the components can be used.
>
> **Further information:** "Application example", Page 998

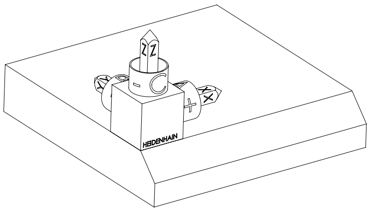
The base vector with components **BX**, **BY** and **BZ** defines the direction of the tilted X axis. The normal vector with components **NX**, **NY** and **NZ** defines the direction of the tilted Z axis and therefore indirectly the working plane. The normal vector is perpendicular to the tilted working plane.

Application example

Example

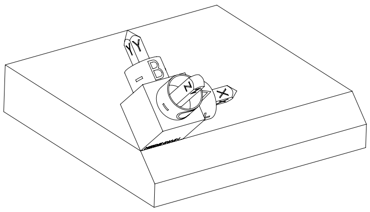
```
11 PLANE VECTOR BX+1 BY+0 BZ+0 NX+0 NY-1 NZ+1 TURN MB MAX FMAX SYM-
   TABLE ROT
```

Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis



Using the defined normal vector with the components **NX+0**, **NY-1** and **NZ+1**, the control orients the Z axis of the working plane coordinate system **WPL-CS** to be perpendicular with the chamfer surface.

The alignment of the tilted X axis equals the orientation of the non-tilted X axis due to component **BX+1**.

The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.

If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following vector components:

- **BX+0**, **BY+1** and **BZ+0** as well as **NX+1**, **NY+0** and **NZ+1** for the second chamfer
- **BX-1**, **BY+0** and **BZ+0** as well as **NX+0**, **NY+1** and **NZ+1** for the third chamfer
- **BX+0**, **BY-1** and **BZ+0** as well as **NX-1**, **NY+0** and **NZ+1** for the fourth chamfer

The values are referenced to the non-tilted workpiece coordinate system **W-CS**.

Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE VECTOR BX+1 BY+0 BZ+0 NX+0 NY-1 NZ+1 TURN MB MAX FMAX SYM-
   TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE VECTOR	Syntax initiator for the working plane definition by means of two vectors
BX, **BY** and **BZ**	Components of base vector, referenced to the workpiece coordinate system **W-CS**, for orienting the tilted X axis Input: **-99.9999999...+99.9999999**
NX, **NY** and **NZ**	Components of the normal vector, referenced to the **W-CS**, for orienting the tilted Z axis Input: **-99.9999999...+99.9999999**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1013
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1016 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1020 Optional syntax element

Notes

- If the components of the normal vector contain very small values, e. g. 0 or 0.0000001, the control cannot determine the working plane slope. In such cases, the control cancels machining with an error message. This behavior cannot be configured.
- The control calculates standardized vectors from the values you enter.

Notes about non-perpendicular vectors

To ensure that the definition of the working plane is unambiguous, the vectors must be programmed perpendicular to each other.

The machine manufacturer uses the optional machine parameter **autoCorrectVector** (no. 201207) to define the behavior of the control with non-perpendicular vectors.

As an alternative to an error message, the control can either correct or replace the non-perpendicular base vector. This correction (or replacement) does not affect the normal vector.

The correction behavior of the control if the base vector is not perpendicular:

- The control projects the base vector along the normal vector onto the working plane defined by the normal vector.

Correction behavior of the control if the base vector is not perpendicular and too short, parallel or antiparallel to the normal vector:

- If the normal vector contains the value 0 in the **NX** component, the base vector corresponds to the original X axis.
- If the normal vector contains the value 0 in the **NY** component, the base vector corresponds to the original Y axis.

Definition

Abbreviation	Definition
B e. g. in **BX**	Base vector
N e. g. in **NX**	Normal vector

PLANE POINTS

Application

Use the **PLANE POINTS** function to define the working plane by three points.

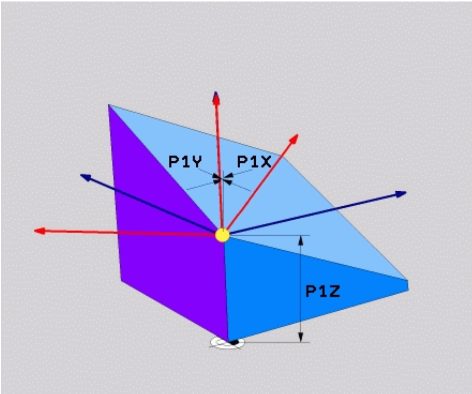
Related topics

- Aligning the plane with touch probe cycle **431 MEASURE PLANE**

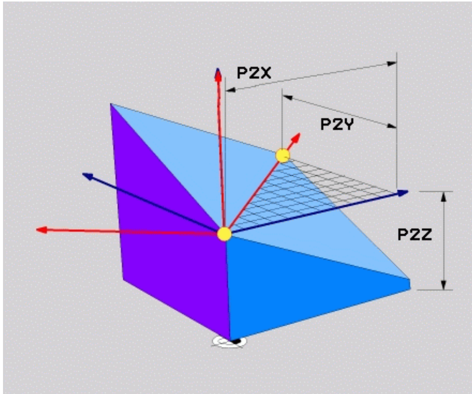
  Further information: "Cycle 431 MEASURE PLANE ", Page 1655

Description of function

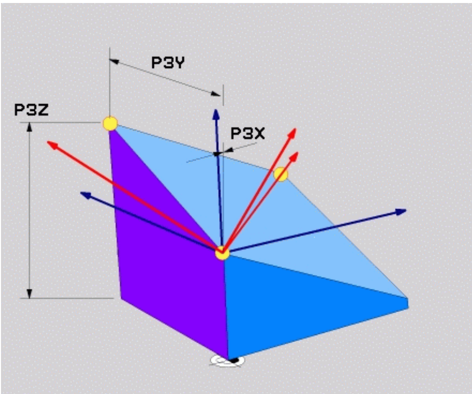
Points define a working plane by using their coordinates in the non-tilted workpiece coordinate system **W-CS**.



First point with coordinates **P1X**, **P1Y** and **P1Z**



Second point with coordinates **P2X**, **P2Y** and **P2Z**



Third point with coordinates **P3X**, **P3Y** and **P3Z**

All nine coordinates must be defined even if one or several coordinates equals 0.

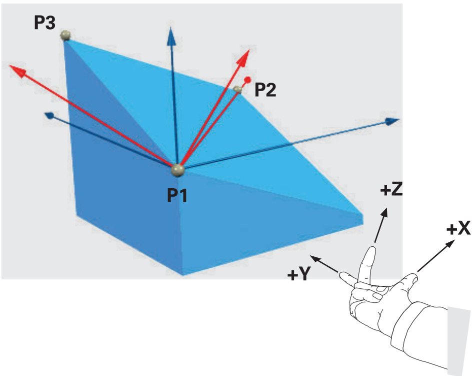
The first point with coordinates **P1X**, **P1Y** and **P1Z** defines the first point of the tilted X axis.

> You can imagine that the first point defines the origin of the tilted X axis and therefore the point serving for orientation of the working plane coordinate system **WPL-CS**.
>
> Ensure that the definition of the first point will not shift the workpiece datum. If the coordinates of the first point are to be programmed with the value 0, the workpiece datum may have to be shifted to that position before.

The second point with coordinates **P2X**, **P2Y** and **P2Z** defines the second point of the tilted X axis and consequently its orientation.

> The orientation of the tilted Y axis in the defined working plane results automatically because both axes are perpendicular to one another.

The third point with coordinates **P3X**, **P3Y** and **P3Z** defines the slope of the tilted working plane.



To direct the positive tool axis direction away from the workpiece, the following conditions apply to the position of the three points:

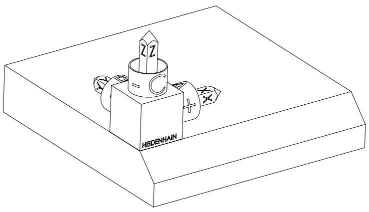
- Point 2 is to the right of point 1
- Point 3 is above the connecting lines between points 1 and 2

Application example

Example

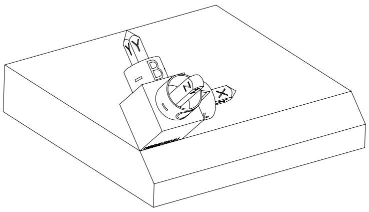
```
11 PLANE POINTS P1X+0  P1Y+0 P1Z+0 P2X+1 P2Y+0 P2Z+0 P3X+0 P3Y+1 P3Z+1
   TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis



Using the first two points **P1** and **P2**, the control orients the X axis of the **WPL-CS**.

The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

P3 defines the slope of the tilted working plane.

The orientations of the tilted Y and Z axes result automatically because all axes are perpendicular to one another.

> The drawing dimensions or any values which will not alter the ratio between the entered values can be used.
>
> In the example, **P2X** may also be defined by the workpiece width **+100**. **P3Y** and **P3Z** can also be programmed by using the chamfer width **+10**.

> When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following points:
>
> - **P1X+0**, **P1Y+0** , **P1Z+0** as well as **P2X+0**, **P2Y+1**, **P2Z+0** and **P3X-1**, **P3Y+0**, **P3Z+1** for the second chamfer
> - **P1X+0**, **P1Y+0** , **P1Z+0** as well as **P2X-1**, **P2Y+0**, **P2Z+0** and **P3X+0**, **P3Y-1**, **P3Z+1** for the third chamfer
> - **P1X+0**, **P1Y+0** , **P1Z+0** as well as **P2X+0**, **P2Y-1**, **P2Z+0** and **P3X+1**, **P3Y+0**, **P3Z+1** for the fourth chamfer
>
> The values are referenced to the non-tilted workpiece coordinate system **W-CS**.
>
> Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE POINTS P1X+0  P1Y+0 P1Z+0 P2X+1 P2Y+0 P2Z+0 P3X+0 P3Y+1 P3Z+1
   TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE POINTS	Syntax initiator for the working plane definition by means of three points
P1X, **P1Y** and **P1Z**	Coordinates of the first point of the tilted X axis, referenced to the workpiece coordinate system **W-CS** Input: **-999999999.999999...+999999999.999999**
P2X, **P2Y** and **P2Z**	Coordinates of the second point, referenced to the **W-CS** for orienting the tilted X axis Input: **-999999999.999999...+999999999.999999**
P3X, **P3Y** and **P3Z**	Coordinates of the third point, referenced to the **W-CS** for inclining the tilted working plane Input: **-999999999.999999...+999999999.999999**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1013
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1016 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1020 Optional syntax element

Definition

Abbreviation	Definition
P e. g. in **P1X**	Point

PLANE RELATIV

Application

Use the **PLANE RELATIV** function to define the working plane by just one spatial angle.

The defined angle always takes effect with reference to the input coordinate system **I-CS**.

Further information: "Reference systems", Page 934

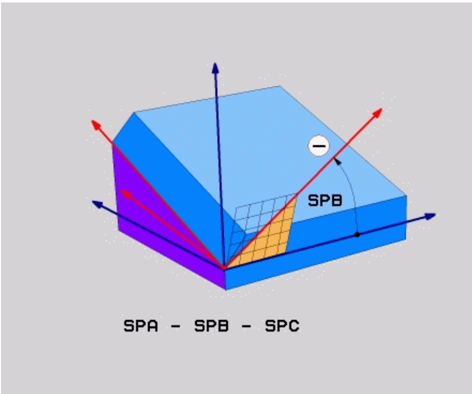
Description of function

A relative spatial angle defines a working plane as a rotation in the active reference system.

When the working plane is not tilted, the defined spatial angle is referenced to the non-tilted workpiece coordinate system **W-CS**.

When the working plane is tilted, the defined spatial angle is referenced to the working plane coordinate system **WPL-CS**.

> **PLANE RELATIV** allows e. g. programming a chamfer on a tilted workpiece surface by tilting the working plane further by the chamfer angle.



Additive spatial angle **SPB**

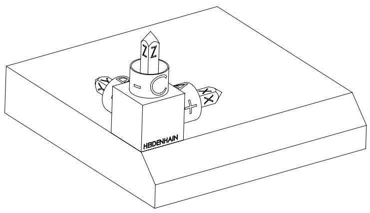
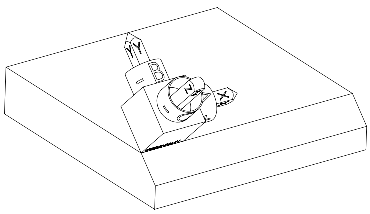
Each **PLANE RELATIV** function defines one spatial angle exclusively. However, it is possible to program any number of **PLANE RELATIV** functions in a row.

If you want to return the working plane that was active before the **PLANE RELATIV** function, define another **PLANE RELATIV** function with the same angle, but with the opposite algebraic sign.

Application example

Example

```
11 PLANE RELATIV SPA+45 TURN MB MAX FMAX SYM- TABLE ROT
```

Initial state 	The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.
Orientation of the tool axis 	Using the spatial angle **SPA+45**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by the **SPA** angle is around the non-tilted X axis. The orientation of the tilted X axis equals the orientation of the non-tilted X axis. The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

> When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following spatial angles:
>
> - First PLANE RELATIVE function with **SPC+90** and another relative tilting with **SPA+45** for the second chamfer
> - First PLANE RELATIVE function with **SPC+180** and another relative tilting with **SPA+45** for the third chamfer
> - First PLANE RELATIVE function with **SPC+270** and another relative tilting with **SPA+45** for the fourth chamfer
>
> The values are referenced to the non-tilted workpiece coordinate system **W-CS**.
>
> Remember that the workpiece datum must be shifted before each working plane definition.

> When shifting the workpiece datum further in a tilted working plane, incremental values must be defined.
>
> **Further information:** "Note", Page 1008

Input

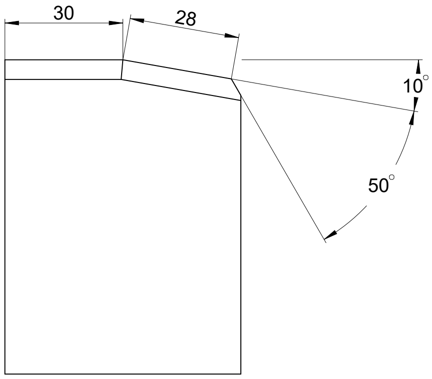
```
11 PLANE RELATIV SPA+45 TURN MB MAX FMAX SYM- TABLE ROT
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE RELATIV	Syntax initiator for the working plane definition by means of one relative spatial angle
SPA, **SPB** or **SPC**	Rotation around the X, Y or Z axis of the workpiece coordinate system **W-CS** Input: **-360.0000000...+360.0000000** When the working plane is tilted, the rotation is effective around the X, Y or Z axis in the working plane coordinate system **WPL-CS**
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1013
SYM or **SEQ**	Select an unambiguous tilting solution **Further information:** "Tilting solution", Page 1016 Optional syntax element
COORD ROT or **TABLE ROT**	Transformation type **Further information:** "Transformation types", Page 1020 Optional syntax element

Note

Incremental datum shift using a chamfer as example



50° chamfer on a tilted workpiece surface

Example

```
11 TRANS DATUM AXIS X+30
12 PLANE RELATIV SPB+10 TURN MB MAX FMAX SYM- TABLE ROT
13 TRANS DATUM AXIS IX+28
14 PLANE RELATIV SPB+50 TURN MB MAX FMAX SYM- TABLE ROT
```

This procedure offers the advantage of being able to program directly with the drawing dimensions.

Definition

Abbreviation	Definition
SP e. g. in **SPA**	Spatial

PLANE RESET

Application

Use the **PLANE RESET** function to reset all tilt angles and deactivate tilting of the working plane.

Description of function

The **PLANE RESET** function always executes two partial tasks:

- Reset all tilt angles, regardless of the selected tilt function or the type of angle
- Deactivate tilting of the working plane

> No other tilting function will carry out this partial task!
> Even when programming all angles with the value 0 in any tilting function, tilting of the working plane remains active.

The optional rotary axis positioning allows tilting the rotary axes back to the home position as the third partial task.

Further information: "Rotary axis positioning", Page 1013

Input

```
11 PLANE RESET TURN MB MAX FMAX
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE RESET	Syntax initiator for resetting all tilting angles and for deactivating an active tilting function
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1013

Note

Before every program run, ensure that no undesired coordinate transformations are effective. When needed, tilting of the working plane can also be deactivated manually in the **3-D rotation** window.

Further information: "3-D rotation window (option 8)", Page 1022

> The status display allows checking the desired status of the tilting situation.
> **Further information:** "Status display", Page 981

PLANE AXIAL

Application

Use the **PLANE AXIAL** function to define the working plane with anywhere from one to three absolute or incremental axis angles.

An axis angle can be programmed for each rotary axis available on the machine.

> Due to the option of defining only one axis angle, **PLANE AXIAL** can also be used on machines with only one rotary axis.

Please note that NC programs with axis angles always depend on the kinematics and therefore depend on the machine in question!

Related topics

- Programming independently of kinematics, using spatial angles

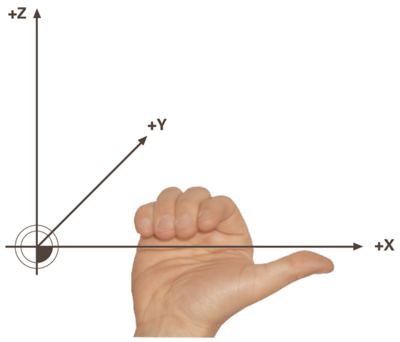
  Further information: "PLANE SPATIAL", Page 984

Description of function

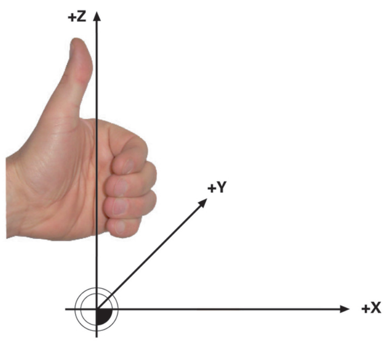
Axis angles define both the orientation of the working plane as well as the nominal coordinates of the rotary axes.

The axis angles must correspond to the axes present on the machine. If you try to program axis angles for rotary axes that do not exist on the machine, the control will generate an error message.

As the axis angles depend on the kinematics, a distinction must be made between the head and the table axes as far as the algebraic signs are concerned.



Extended right-hand rule for head rotary axes



Extended left-hand rule for table rotary axes

The thumb of the hand in question points in the positive direction of the axis around which the rotation occurs. If you curl your fingers, the curled fingers point in the positive direction of rotation.

Bear in mind that when working with rotary axes layered on top of one another, the positioning of the first rotary axis will also modify the position of the second rotary axis.

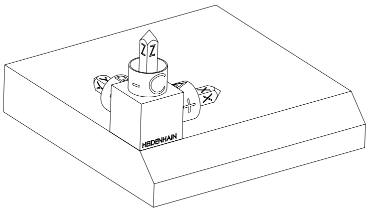
Application example

The example below applies to a machine with AC table kinematics whose two rotary axes are perpendicular and layered on top of one another.

Example

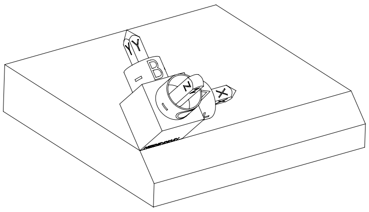
```
11 PLANE AXIAL A+45 TURN MB MAX FMAX
```

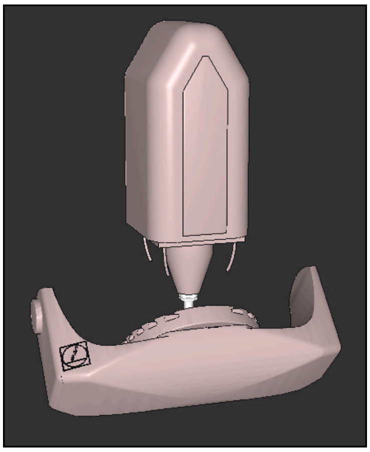
Initial state



The initial state shows the position and orientation of the working plane coordinate system **WPL-CS** while still non-tilted. The workpiece datum which in the example was shifted to the top chamfer edge defines the position. The active workpiece datum also defines the position around which the control orients or rotates the **WPL-CS**.

Orientation of the tool axis





Using the defined axis angle **A**, the control orients the Z axis of the **WPL-CS** to be perpendicular with the chamfer surface. The rotation by angle **A** is around the non-tilted X axis.

> To position the tool perpendicular to the chamfer surface, table rotary axis A must tilt to the rear.
>
> In accordance with the extended left-hand rule for table axes, the algebraic sign of the A axis value must be positive.

The orientation of the tilted X axis equals the orientation of the non-tilted X axis.

The orientation of the tilted Y axis results automatically because all axes are perpendicular to one another.

> When programming the machining of the chamfer within a subprogram, an all-round chamfer can be produced using four working plane definitions.
>
> If the example defines the working plane of the first chamfer, the remaining chamfers can be programmed using the following axis angles:
>
> - **A+45** and **C+90** for the second chamfer
> - **A+45** and **C+180** for the third chamfer
> - **A+45** and **C+270** for the fourth chamfer
>
> The values are referenced to the non-tilted workpiece coordinate system **W-CS**.
>
> Remember that the workpiece datum must be shifted before each working plane definition.

Input

```
11 PLANE AXIAL A+45 TURN MB MAX FMAX
```

The NC function includes the following syntax elements:

Syntax element	Meaning
PLANE AXIAL	Syntax initiator for the working plane definition using one to three axis angle
A	When an A axis is available, nominal position of the A rotary axis Input: **-99999999.9999999...+99999999.9999999** Optional syntax element
B	When a B axis is available, nominal position of the B rotary axis Input: **-99999999.9999999...+99999999.9999999** Optional syntax element
C	When a C axis is available, nominal position of the C rotary axis Input: **-99999999.9999999...+99999999.9999999** Optional syntax element
MOVE, **TURN** or **STAY**	Type of rotary axis positioning Depending on the selection, the optional syntax elements **MB**, **DIST** and **F**, **F AUTO** or **FMAX** can be defined. **Further information:** "Rotary axis positioning", Page 1013

The **SYM** or **SEQ** entries as well as **COORD ROT** or **TABLE ROT** are possible, but are not effective in conjunction with **PLANE AXIAL**.

Notes

Refer to your machine manual.

If your machine allows spatial angle definitions, you can continue your programming with **PLANE RELATIV** after **PLANE AXIAL**.

- The axis angles of the **PLANE AXIAL** function are modally effective. If you program an incremental axis angle, the control will add this value to the currently effective axis angle. If you program two different rotary axes in two successive **PLANE AXIAL** functions, the new working plane is derived from the two defined axis angles.
- The **PLANE AXIAL** function does not take basic rotation into account.
- When used in conjunction with **PLANE AXIAL**, the programmed transformations mirroring, rotation and scaling do not affect the position of the rotation point nor the orientation of the rotary axes.

  Further information: "Transformations in the workpiece coordinate system (W-CS)", Page 940
- Without the use of a CAM system, **PLANE AXIAL** is convenient only with rotary axes positioned at right angles.

Rotary axis positioning

Application

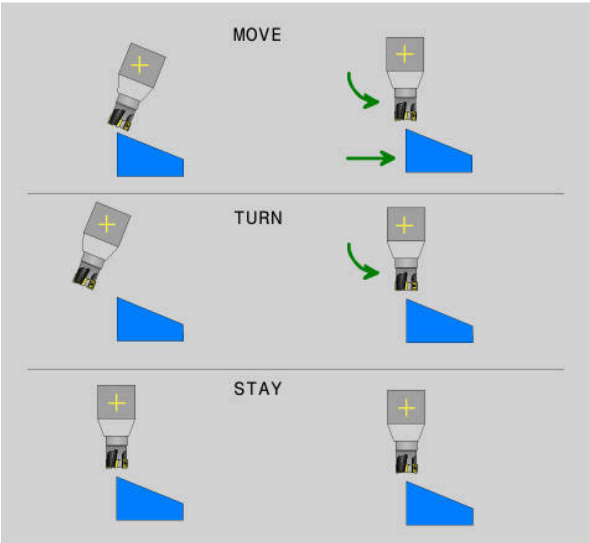
The type of rotary axis positioning defines how the control tilts the rotary axes to the calculated axis values.

The selection depends e. g. on the aspects below:

- Is the tool near the workpiece during tilting to position?
- Is the tool at a safe tilting position during tilting to position?
- May and can the rotary axes be positioned automatically?

Description of function

The control offers three types of rotary axis positioning from which one must be selected.



Type of rotary axis positioning	Meaning
MOVE	If you perform tilting near the workpiece, then use this option. **Further information:** "Rotary axis positioning MOVE", Page 1014
TURN	If the workpiece is so large that the range of traverse is not sufficient for the compensating movement of the linear axes, then use this option. **Further information:** "Rotary axis positioning TURN", Page 1014
STAY	The control does not position any axes. **Further information:** "Rotary axis positioning STAY", Page 1015

Rotary axis positioning MOVE

The control positions the rotary axes and performs compensation movements in the linear principal axes.

The compensation movements ensure that the relative position between the tool and the workpiece will not change during the positioning process.

NOTICE

Danger of collision!

The center of rotation is in the tool axis. In the case of large tool diameters, the tool may plunge into the material during tilting. During the tilting movement, there is a risk of collision!

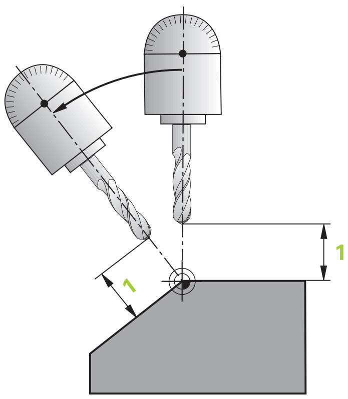
- Ensure sufficient distance between the tool and the workpiece

When **DIST** is not defined or when defining the value 0, the center of rotation and consequently the center of the compensation movements is in the tool tip.

When defining **DIST** with a value greater than 0, the center of rotation in the tool axis is shifted away from the tool tip by this value.

When wishing to tilt around a certain point on the workpiece, ensure the following:

- Prior to tilting to position, the tool is positioned directly above the desired point on the workpiece.
- The value defined in **DIST** matches exactly the clearance between the tool tip and the desired center of rotation.



Rotary axis positioning TURN

The control positions only the rotary axes. The tool must be positioned after tilting to position.

Rotary axis positioning STAY

Both the rotary axes and the tool must be positioned after tilting to position.

> Even with **STAY**, the control orients the working plane coordinate system **WPL-CS** automatically.

When selecting **STAY**, the rotary axes must be tilted to position in a separate positioning block after the **PLANE** function.

In the positioning block, use only the axis angles calculated by the control:

- **Q120** for the axis angle of the A axis
- **Q121** for the axis angle of the B axis
- **Q122** for the axis angle of the C axis

The variable avoids entry and calculating errors. In addition, no changes are required after changing the values within the **PLANE** functions.

Example

```
11 L A+Q120 C+Q122 FMAX
```

Input

MOVE

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 MOVE DIST0 FMAX
```

Selecting **MOVE** allows defining the syntax elements below:

Syntax element	Meaning
DIST	Distance between center of rotation and the tool tip Input: **0...99999999.9999999** Optional syntax element
F, **F AUTO** or **FMAX**	Feed rate definition for automatic rotary axis positioning Optional syntax element

TURN

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX
```

Selecting **TURN** allows defining the syntax elements below:

Syntax element	Meaning
MB	Retraction in the current tool axis direction before positioning the rotary axis Values with an incremental effect can be entered or a retraction up to the traverse limit can be defined by selecting **MAX**. Input: **0...99999999.9999999** or **MAX** Optional syntax element
F, **F AUTO** or **FMAX**	Feed rate definition for automatic rotary axis positioning Optional syntax element

STAY

```
11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 TURN MB MAX FMAX
```

Selecting **STAY** does not allow defining further syntax elements.

Note

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect or no pre-positioning before tilting the tool into position can lead to a risk of collision during the tilting movement!

- Program a safe position before the tilting movement
- Carefully test the NC program or program section in the **Program run, single block** operating mode

Tilting solution

Application

SYM (SEQ) allows selecting the desired option from several tilting solutions.

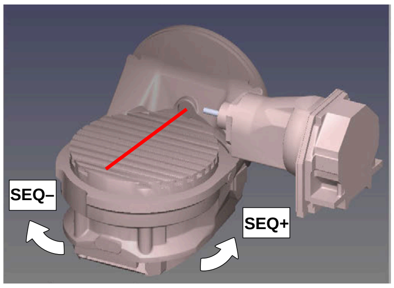
Unambiguous tilting solutions can be defined by using axis angles exclusively.

All other definition options can result in several tilting solutions, depending on the machine.

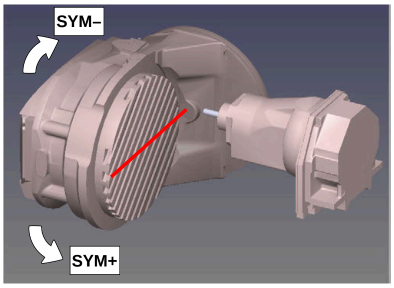
Description of function

The control offers two options from which one must be selected.

Option	Meaning
SYM	With **SYM** you select a tilting solution relative to the symmetry point of the master axis. **Further information:** "Tilting solution SYM", Page 1017
SEQ	With **SEQ** you select a tilting solution relative to the basic position of the master axis. **Further information:** "Tilting solution SEQ", Page 1018



Reference for **SEQ**



Reference for **SYM**

If the solution you have selected with **SYM** (**SEQ**) is not within the machine's range of traverse, then the control displays the **Entered angle not permitted** error message.

The entry of **SYM** or **SEQ** is optional.

If you do not define **SYM** (**SEQ**), then the control determines the solution as follows:

1 Check whether both possible solutions are within the traverse range of the rotary axes
2 Two possible solutions: Based on the current position of the rotary axes, choose the possible solution with the shortest path
3 One possible solution: Choose the only solution
4 No possible solution: Issue the error message **Entered angle not permitted**

Tilting solution SYM

With the **SYM** function, you select one of the possible solutions relative to the symmetry point of the master axis:

- **SYM+** positions the master axis in the positive half-space relative to the symmetry point
- **SYM-** positions the master axis in the negative half-space relative to the symmetry point

As opposed to **SEQ**, **SYM** uses the symmetry point of the master axis as its reference. Every master axis has two symmetry positions, which are 180° apart from each other (sometimes only one symmetry position is in the traverse range).

To determine the symmetry point:

- Perform **PLANE SPATIAL** with any spatial angle and **SYM+**
- Save the axis angle of the master axis in a Q parameter (e.g., –80)
- Repeat the **PLANE SPATIAL** function with **SYM-**
- Save the axis angle of the master axis in a Q parameter (e.g., –100)
- Calculate the average value (e.g., –90)

The average value corresponds to the symmetry point.

Tilting solution SEQ

With the **SEQ** function, you select one of the possible solutions relative to the home position of the master axis:

- **SEQ+** positions the master axis in the positive tilting range relative to the home position
- **SEQ-** positions the master axis in the negative tilting range relative to the home position

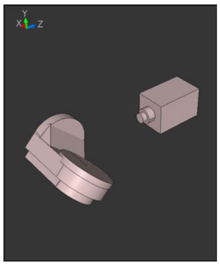
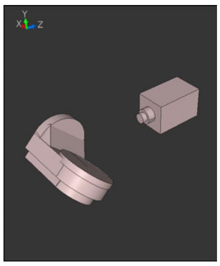
SEQ assumes that the master axis is in its home position (0°). Relative to the tool, the master axis is the first rotary axis, or the last rotary axis relative to the table (depending on the machine configuration). If both possible solutions are in the positive or negative range, then the control automatically uses the closer solution (shorter path). If you need the second possible solution, then you must either pre-position the master axis (in the area of the second possible solution) before tilting the working plane, or work with **SYM**.

Examples

Machine with C rotary axis and A tilting table. Programmed function: PLANE SPATIAL SPA+0 SPB+45 SPC+0

Limit switch	Start position	SYM = SEQ	Resulting axis position
None	A+0, C+0	Not prog.	A+45, C+90
None	A+0, C+0	+	A+45, C+90
None	A+0, C+0	–	A−45, C−90
None	A+0, C−105	Not prog.	A−45, C−90
None	A+0, C−105	+	A+45, C+90
None	A+0, C−105	–	A−45, C−90
−90 < A < +10	A+0, C+0	Not prog.	A−45, C−90
−90 < A < +10	A+0, C+0	+	Error message
−90 < A < +10	A+0, C+0	–	A−45, C−90

Machine with B rotary axis and A tilting table (limit switches: A +180 and –100). Programmed function: PLANE SPATIAL SPA-45 SPB+0 SPC+0

SYM	SEQ	Resulting axis position	Kinematics view
+		A−45, B+0	
-		Error message	**No solution in limited range**
	+	Error message	**No solution in limited range**
	-	A−45, B+0	

> The position of the symmetry point is contingent on the kinematics. If you change the kinematics (such as switching the head), then the position of the symmetry point changes as well.
>
> Depending on the kinematics, the positive direction of rotation of **SYM** may not correspond to the positive direction of rotation of **SEQ**. Therefore, ascertain the position of the symmetry point and the direction of rotation of **SYM** on each machine before programming.

Transformation types

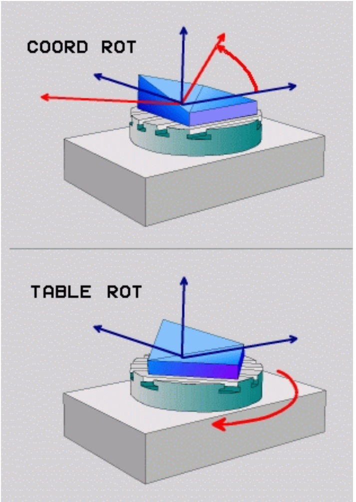
Application

COORD ROT and **TABLE ROT** influence the orientation of the working plane coordinate system **WPL-CS** through the axis position of a free rotary axis.

> Any rotary axis becomes a free rotary axis with the following configuration:
>
> - The rotary axis has no effect on the tool angle of inclination because the rotary axis and the tool axis are parallel in the tilting situation
> - The rotary axis is the first rotary axis in the kinematic chain starting from the workpiece
>
> The effect of the **COORD ROT** and **TABLE ROT** transformation types therefore depends on the programmed spatial angles and the machine kinematics.

Description of function

The control offers two options.



Option	Meaning
COORD ROT	> The control positions the free rotary axis to 0 > The control orients the working plane coordinate system in accordance with the programmed spatial angle
TABLE ROT	**TABLE ROT** with: ■ SPA **and** SPB **equal to** 0 ■ SPC **equal or unequal to** 0 > The control orients the free rotary axis in accordance with the programmed spatial angle > The control orients the working plane coordinate system in accordance with the basic coordinate system **TABLE ROT** with: ■ **At least** SPA **or** SPB **unequal to** 0 ■ SPC **equal or unequal to** 0 > The control does not position the free rotary axis. The position prior to tilting the working plane is maintained > Since the workpiece was not positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle

If no free rotary axis arises in a tilting situation, then the **COORD ROT** and **TABLE ROT** transformation types have no effect.

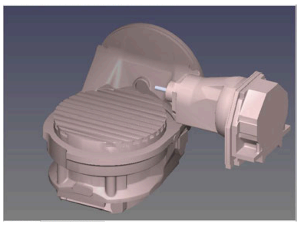
The entry of **COORD ROT** or **TABLE ROT** is optional.

If no transformation type was selected, then the control uses the **COORD ROT** transformation type for the **PLANE** functions

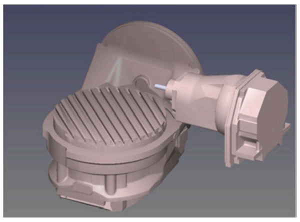
Example

The following example shows the effect of the **TABLE ROT** transformation type in conjunction with a free rotary axis.

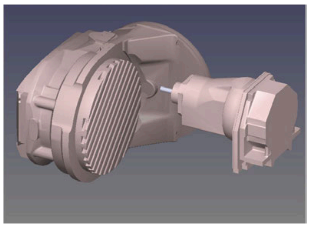
11 L B+45 R0 FMAX	; Pre-position the rotary axis
12 PLANE SPATIAL SPA-90 SPB+20 SPC +0 TURN F5000 TABLE ROT	; Tilt the working plane



Origin



A = 0, B = 45



A = −90, B = 45

> The control positions the B axis to the axis angle B+45
> With the programmed tilting situation with SPA−90, the B axis becomes the free rotary axis
> The control does not position the free rotary axis. The position of the B axis prior to the tilting of the working plane is maintained
> Since the workpiece was not also positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle SPB +20

Notes

- For the positioning behavior with the **COORD ROT** and **TABLE ROT** transformation types, it makes no difference whether the free rotary axis is a table axis or a head axis.
- The resulting axis position of the free rotary axis depends on an active basic rotation, among other factors.
- The orientation of the working plane coordinate system also depends on a programmed rotation; for example, by means of Cycle **10 ROTATION**.

15.7.3 3-D rotation window (option 8)

Application

The **3-D rotation** window allows activating and deactivating tilting of the working plane for the **Manual** and **Program Run** operating modes. This allows restoring the tilted working plane and retracting the tool, e. g. after program cancellation in the **Manual operation** application.

Related topics

- Tilting the working plane in the NC program

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 979
- Reference systems of the control

  Further information: "Reference systems", Page 934

Requirements

- Machine with rotary axes
- Kinematics description

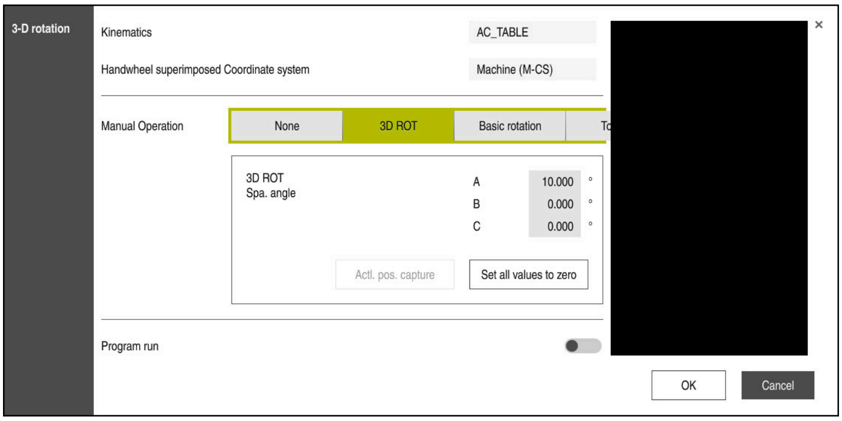
  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 1 (software option 8)
- Function enabled by the machine manufacturer

  In the machine parameter **rotateWorkPlane** (no. 201201), the machine manufacturer defines whether tilting the working plane is allowed on the machine.
- Tool with tool axis **Z**

Description of function

The **3-D rotation** window can be opened with the **3D ROT** button in the **Manual operation** application.

Further information: "Manual operation application", Page 180



3-D rotation window

The **3-D rotation** window contains the following information:

Area	Contents
Kinematics	Name of the active machine kinematics
Handwheel superimpositioning coordinate system	Coordinate system in which handwheel superimpositioning is effective **Further information:** "Reference systems", Page 934 **Further information:** "Function Handwheel superimp.", Page 1141 **Further information:** "Activating handwheel superimpositioning with M118", Page 1239

Area	Contents
Manual Operation	Status of tilting function in the **Manual** operating mode: ■ **None** The control will not take the rotary axes positions that are not equal to 0 into account. Traverses are effective in the **W-CS** workpiece coordinate system. **Further information:** "Workpiece coordinate system W-CS", Page 940 ■ **3D ROT** The control takes the positions of rotary axes and columns **SPA**, **SPB** and **SPC** of the preset table into account. The traverses are effective in the **WPL-CS** working plane coordinate system. **Further information:** "3D ROT setting", Page 1024 ■ **Basic rotation** The control takes the columns **SPA**, **SPB** and **SPC** into account, but no rotary axis positions that are not equal to 0. Traverses are effective in the **W-CS** workpiece coordinate system. **Further information:** "Basic rotation setting", Page 1025 ■ **Tool axis** This is relevant only for head rotary axes. The traverses are effective in the **T-CS** tool coordinate system. **Further information:** "Tool axis setting", Page 1025
Program run	When activating the **Tilt working plane** function for the **Program run** operating mode, the entered angle of rotation applies from the first NC block of the NC program to be run. If you use Cycle **19 WORKING PLANE** or the **PLANE** function in the NC program, the angular values defined there become effective. The control will reset the entered angular values to 0.

Confirm the settings with **OK**.

When a tilting function is active in the **3-D rotation** window, the control displays the corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 141

3D ROT setting

When selecting the **3D ROT** setting, all axes move in the tilted working plane. The traverses are effective in the **WPL-CS** working plane coordinate system.

Further information: "Working plane coordinate system WPL-CS", Page 942

If a basic rotation or 3-D basic rotation has additionally been saved to the preset table, it will automatically be taken into account.

The control displays the angles that are currently effective in the entry field of the **Manual Operation** area. The spatial angle can also be edited.

The **3D ROT** entry field in the **Manual Operation** area displays the currently active angles. The machine manufacturer uses the machine parameter **planeOrientation** (no. 201202) to define whether the control calculates with spatial angles **SPA**, **SPB** and **SPC** or with the axis values of the existing rotary axes.

> **i** When editing values in the **3D ROT** entry field, the rotary axes must be positioned afterwards, e. g. in the **MDI** application.

Basic rotation setting

When selecting the **Basic rotation** setting, the axes move while taking a basic rotation or 3D basic rotation into account.

Further information: "Basic rotation and 3D basic rotation", Page 951

The axis movements are effective in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 940

If the active workpiece preset contains a basic rotation or 3D basic rotation, the control additionally displays the corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 141

The **3D ROT** entry field has no function in this setting.

Tool axis setting

When selecting the **Tool axis** setting, traversing is possible in both the positive and negative direction of the tool axis. The control locks all other axes. This setting makes sense only for machines with head rotary axes.

The traverse is effective in the **T-CS** tool coordinate system.

Further information: "Tool coordinate system T-CS", Page 946

This setting is used e. g. in the cases below:

- When retracting the tool in the direction of the tool axis during an interruption of a 5-axis machining program.
- When traversing with the axis keys or the handwheel with a pre-positioned tool.

The **3D ROT** entry field has no function in this setting.

Notes

- The control uses the **COORD ROT** transformation type in the following situations:
  - if a **PLANE** function was previously executed with **COORD ROT**
  - after **PLANE RESET**
  - with corresponding configuration of the machine parameter **CfgRotWorkPlane** (no. 201200) by the machine manufacturer
- The control uses the **TABLE ROT** transformation type in the following situations:
  - if a **PLANE** function was previously executed with **TABLE ROT**
  - with corresponding configuration of the machine parameter **CfgRotWorkPlane** (no. 201200) by the machine manufacturer
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (option 8). If the rotary axes are positioned so that they differ from the definition in the **3-D rotation** window, the control cancels this process with an error message by default.

  In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.
- A tilted working plane will remain active even after a control restart.

  Further information: "Referencing workspace", Page 174
- PLC positionings defined by the machine manufacturer are not allowed when the working plane is tilted.

15.8 Inclined machining (option 9)

Application

When pre-positioning the tool during machining, workpiece positions that are difficult to reach can be machined without collisions.

Related topics

- Compensating the tool angle of inclination with **FUNCTION TCPM** (option 9)

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027
- Compensating the tool angle of inclination with **M128** (option 9)

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1245
- Tilting the working plane (option 8)

  Further information: "Tilting the working plane (option 8)", Page 978
- Presets on the tool

  Further information: "Presets on the tool", Page 244
- Reference systems

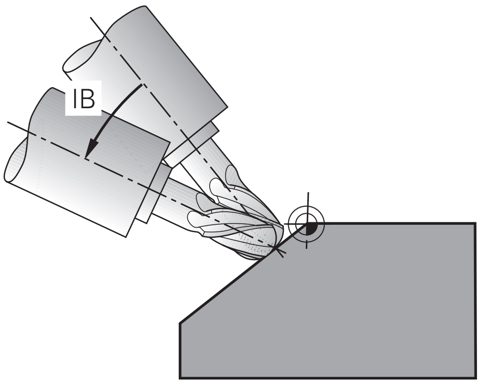
  Further information: "Reference systems", Page 934

Requirements

- Machine with rotary axes
- Kinematics description

  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 2 (software option 9)

Description of function



The **FUNCTION TCPM** function allows executing inclined machining. In this process, one working plane may be tilted.

Further information: "Tilting the working plane (option 8)", Page 978

Inclined machining can be implemented using the following functions:

- Incremental traverse of rotary axis

  Further information: "Inclined machining with incremental process", Page 1026
- Normal vectors

  Further information: "Inclined machining using normal vectors", Page 1027

Inclined machining with incremental process

Inclined machining can be implemented by changing the inclination angle in addition to the normal linear movement while function **FUNCTION TCPM** or **M128** is active, e. g. **L X100 Y100 IB-17 F1000 G01 G91 X100 Y100 IB-17 F1000**. In this process, the relative position of the tool's center of rotation remains the same while inclining the tool.

Example

* - ...	
12 L Z+50 R0 FMAX	; Position at clearance height
13 PLANE SPATIAL SPA+0 SPB-45 SPC +0 MOVE DIST50 F1000	; Define and activate the PLANE function
14 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS	; Activate TCPM
15 L IB-17 F1000	; Pre-position the tool
* - ...	

Inclined machining using normal vectors

In case of inclined machining using normal vectors, the tool angle of inclination is achieved by means of straight lines **LN**.

To execute inclined machining with normal vectors, function **FUNCTION TCPM** or miscellaneous function **M128** must be activated.

Example

* - ...	
12 L Z+50 R0 FMAX	; Position at clearance height
13 PLANE SPATIAL SPA+0 SPB+45 SPC +0 MOVE DIST50 F1000	; Tilt the working plane
14 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS	; Activate TCPM
15 LN X+31.737 Y+21,954 Z+33,165 NX+0,3 NY+0 NZ+0,9539 F1000 M3	; Incline the tool with the normal vector
* - ...	

15.9 Compensating the tool angle of inclination with FUNCTION TCPM (option 9)

Application

The **FUNCTION TCPM** function allows you to influence the positioning behavior of the control. When activating **FUNCTION TCPM**, the control compensates any changed tool angles of inclination by means of compensating movements of the linear axes.

FUNCTION TCPM allows e. g. changing the tool angle of inclination for inclined machining while the position of the tool location point relative to the contour remains the same.

> Instead of **M128**, HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM**.

Related topics

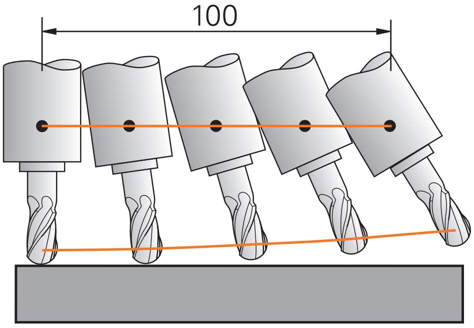
- Compensating the tool angle of inclination with **M128**
  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1245
- Tilting the working plane
  Further information: "Tilting the working plane (option 8)", Page 978
- Presets on the tool
  Further information: "Presets on the tool", Page 244
- Reference systems
  Further information: "Reference systems", Page 934

Requirements

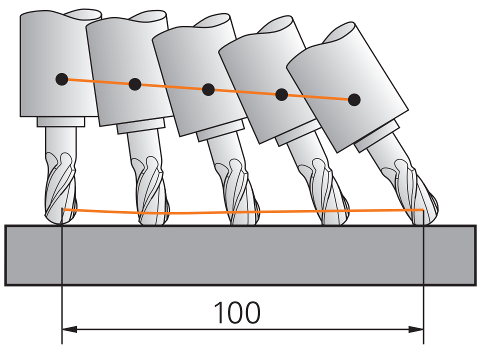
- Machine with rotary axes
- Kinematics description
  To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 2 (software option 9)

Description of function

FUNCTION TCPM is an improvement on the **M128** function which allows defining the behavior of the control while during the positioning of rotary axes.



Behavior without **TCPM**



Behavior with **TCPM**

When **FUNCTION TCPM** is active, the control shows the **TCPM** icon in the position display.

Further information: "Positions workspace", Page 141

The **FUNCTION RESET TCPM** function resets the **FUNCTION TCPM** function.

Input

FUNCTION TCPM

```
10 FUNCTION TCPM F TCP AXIS POS PATHCTRL AXIS REFPNT CENTER-CENTER F1000
```

The NC function contains the following syntax elements:

Syntax element	Meaning
FUNCTION TCPM	Syntax initiator for compensating tool angles of inclination
F TCP or **F CONT**	Interpretation of the programmed feed rate **Further information:** "Interpretation of the programmed feed rate ", Page 1029
AXIS POS or **AXIS SPAT**	Interpretation of programmed rotary axis coordinates **Further information:** "Interpretation of the programmed rotary axis coordinates", Page 1030
PATHCTRL AXIS or **PATHCTRL VECTOR**	Interpolation of tool angle of inclination **Further information:** "Interpolation of tool angle of inclination between start and end positions", Page 1031
REFPNT TIP-TIP, **REFPNT TIP-CENTER** or **REFPNT CENTER-CENTER**	Selection of tool location point and tool rotation point **Further information:** "Selection of tool location point and tool rotation point", Page 1032 Optional syntax element
F	Maximum feed rate for compensating movements in the linear axes for movements with a rotary-axis component **Further information:** "Limiting the linear-axis feed rate", Page 1033 Optional syntax element

FUNCTION RESET TCPM

```
10 FUNCTION RESET TCPM
```

The NC function contains the following syntax elements:

Syntax element	Meaning
FUNCTION RESET TCPM	Syntax initiator for resetting of **FUNCTION TCPM**

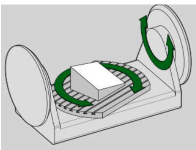
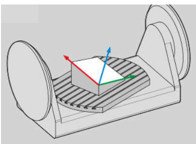
Interpretation of the programmed feed rate

The control offers the following options for interpreting the feed rate:

Selection	Function
F TCP	When selecting **F TCP**, the control interprets the programmed feed rate as the relative speed between the tool location point and the workpiece.
F CONT	When selecting **F CONT**, the control interprets the programmed feed rate as contouring feed rate. In this process, the control transfers the contouring feed rate to the respective axes of the active NC block.

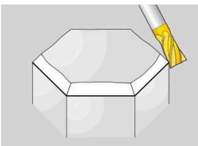
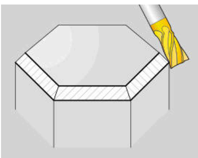
Interpretation of the programmed rotary axis coordinates

The control offers the options below for interpreting the tool angle of inclination between the start and end position:

Selection	Function
 AXIS POS	When selecting **AXIS POS**, the control interprets the programmed rotary axis coordinates as axis angle. The control positions the rotary axes on the position defined in the NC program. The **AXIS POS** selection is primarily suitable in conjunction with perpendicularly arranged rotary axes. **AXIS POS** can only be used with different machine kinematics, e. g. 45° swivel heads if the programmed rotary axis coordinates define the desired working plane alignment correctly, e. g. using a CAM system.
 AXIS SPAT	If **AXIS SPAT** is selected, the control interprets the programmed rotary axis coordinates as spatial angles. The control preferably implements the spatial angles as orientation of the coordinate system and tilts only required axes. Select **AXIS SPAT** to allow using NC programs regardless of kinematics. Selecting **AXIS SPAT** allows defining spatial angles that are given with respect to the input coordinate system **I-CS**. The defined angles have the effect of incremental spatial angles. In the first traverse block after the **FUNCTION TCPM** function with **AXIS SPAT**, always program **SPA**, **SPB** and **SPC**, even with spatial angles of 0°. **Further information:** "Input coordinate system I-CS", Page 944

Interpolation of tool angle of inclination between start and end positions

The control offers the options below for interpolating the tool angle of inclination between the programmed start and end positions:

Selection	Function
 **PATHCTRL AXIS**	When selecting **PATHCTRL AXIS**, the control interpolates linearly between the start and end point. Use **PATHCTRL AXIS** with NC programs with small changes of the tool angle of inclination per NC block. In this case, the angle **TA** in Cycle **32** can be large. **Further information:** "Cycle 32 TOLERANCE ", Page 1128 **PATHCTRL AXIS** can be used both for face milling and also for peripheral milling. **Further information:** "3D tool compensation during face milling (option 9)", Page 1053 **Further information:** "3D tool compensation during peripheral milling (option 9)", Page 1058
 **PATHCTRL VECTOR**	If **PATHCTRL VECTOR** is selected, the tool orientation within an NC block always lies in the plane that is defined by the start orientation and end orientation. With **PATHCTRL VECTOR** the control generate a plane surface even if there are large changes in the tool inclination angle. Use **PATHCTRL VECTOR** for peripheral milling if there are large changes in the tool inclination angle per NC block.

In both cases, the control moves the programmed tool location point on a straight line between the start position and end position.

> To obtain continuous movement, define Cycle **32** with a **tolerance for rotary axes**.
> **Further information:** "Cycle 32 TOLERANCE ", Page 1128

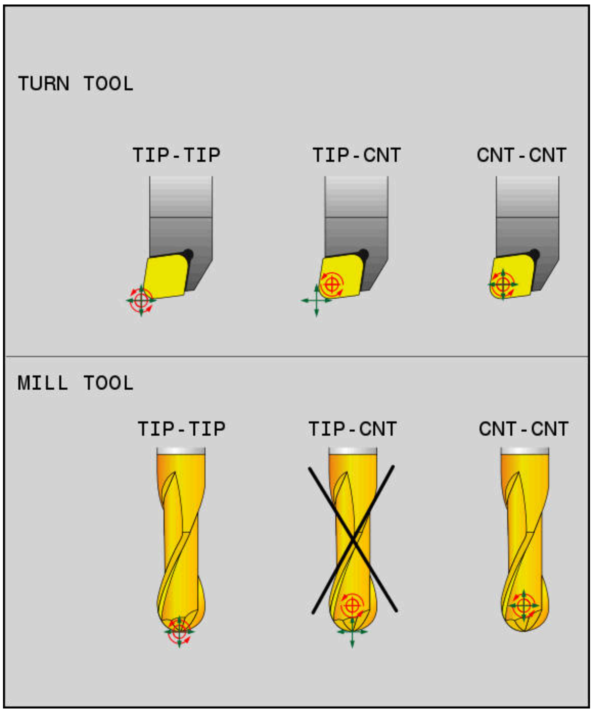
Selection of tool location point and tool rotation point

The control offers the options below for defining the tool location point and the tool rotation point:

Selection	Function
REFPNT TIP-TIP	When selecting **REFPNT TIP-TIP**, the tool location point and the tool rotation point are located at the tool tip.
REFPNT TIP-CENTER	When selecting **REFPNT TIP-CENTER**, the tool location point is located at the tool tip. The tool rotation point is located at the tool center point. The option **REFPNT TIP-CENTER** is optimized for turning tools (option 50). When the control positions the rotary axes, the tool rotation point remains at the same position. This allows you to machine e. g. complex contours by simultaneous turning. **Further information:** "Theoretical and virtual tool tip", Page 1042
REFPNT CENTER-CENTER	When selecting **REFPNT CENTER-CENTER**, the tool location point and the tool rotation point are located at the tool center point. Selecting **REFPNT CENTER-CENTER** allows executing CAM-generated NC programs which are referenced to the tool center point and still calibrate the tool relative to its tip. This allows the control to monitor the entire tool length for collisions while machining is in progress. Previously, this functionality could only be achieved by shortening the tool with **DL** and without the control monitoring the remaining tool length. **Further information:** "Tool data within variables", Page 1038 If you use **REFPNT CENTER-CENTER** to program pocket milling cycles, the control generates an error message. **Further information:** "Overview", Page 452

Further information: "Presets on the tool", Page 244

The reference point is optional. If you do not enter anything, the control uses **REFPNT TIP-TIP**.



Selection options of tool preset and tool rotation point

Limiting the linear-axis feed rate

The optional input of **F** allows you to limit the feed rate of linear axes for motions with a rotary-axis component.

Thus, you can avoid fast compensation movements, e.g. in case of retraction movement at rapid traverse.

> Make sure to select a value for the linear axis feed-rate limit that is not too small because large feed-rate variations may occur at the tool location point. Feed-rate variations impair the surface quality.
>
> If **FUNCTION TCPM** is active, the feed-rate limit will only be effective for motions with a rotary-axis component, not for entirely linear motions.

The linear axis feed-rate limit remains in effect until you program a new value or reset **FUNCTION TCPM**.

Notes

NOTICE

Danger of collision!

Rotary axes with Hirth coupling must move out of the coupling to enable tilting. There is a danger of collision while the axis moves out of the coupling and during the tilting operation.

- Retract the tool before changing the position of the tilting axis

- Before positioning axes with **M91** or **M92,** and before a **TOOL CALL** block, reset the **FUNCTION TCPM** function.
- The following cycles can be used with active **FUNCTION TCPM**:
  - Cycle **32 TOLERANCE**
  - Cycle **800 ADJUST XZ SYSTEM** (option 50)
  - Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** (option 158)
  - Cycle **883 TURNING SIMULTANEOUS FINISHING** (option 158)
  - Cycle **444 PROBING IN 3-D**
- Use only ball-nose cutters for face milling in order to avoid contour damage. In combination with other tool shapes, check if the NC program contains any possible contour damage, using the **Simulation** workspace.

  Further information: "Notes", Page 1249

16

Compensations

16.1 Tool compensation for tool length and radius

Application

Delta values allow implementing tool compensation of the tool length and the tool radius. Delta values influence the calculated and therefore the active tool dimensions.

The tool length delta value **DL** is effective in the tool axis. The tool radius delta value **DR** is effective exclusively for radius-compensated traverses with the path functions and cycles.

Further information: "Path Functions", Page 289

Related topics

- Tool radius compensation

  Further information: "Tool radius compensation", Page 1038
- Tool compensation with compensation tables

  Further information: "Tool compensation with compensation tables", Page 1044

Description of function

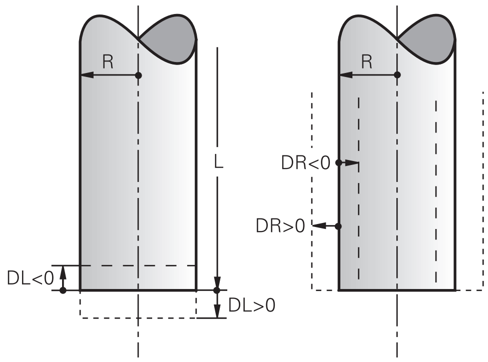
The control distinguishes between two types of delta values:

- Delta values within the tool table serve for permanent tool compensation that is required e. g. due to wear.

  These delta values can be determined e. g. by using a tool touch probe. The control automatically enters the delta values in the tool management.

  Further information: "Tool management ", Page 270
- Delta values within a tool call serve for a tool compensation that is effective exclusively in the current NC program, e. g. a workpiece oversize.

  Further information: "Tool call by TOOL CALL", Page 277



Delta values represent deviations from the length and radius of a tool.

A positive delta value enlarges the current tool length or the tool radius. The tool then cuts less material during machining, e. g. for a workpiece oversize.

A negative delta value reduces the current tool length or the tool radius. The tool then cuts more material during machining.

For programming delta values in an NC program, define the value within a tool call or by using a compensation table.

Further information: "Tool call by TOOL CALL", Page 277

Further information: "Tool compensation with compensation tables", Page 1044

Delta values within a tool call can also be defined by using variables.

Further information: "Tool data within variables", Page 1038

Tool length compensation

The control takes the tool length compensation into account as soon as a tool is called. The control performs tool length compensation only on tools of length L>0.

In tool length compensation, the control takes delta values from the tool table and the NC program into account.

Active tool length = **L** + $\mathbf{DL}_{TAB}$ + $\mathbf{DL}_{Prog}$

L:	Tool length **L** from the tool table **Further information:** "Tool table tool.t", Page 1814
$\mathbf{DL}_{TAB}$:	Tool length delta value **DL** from the tool table **Further information:** "Tool table tool.t", Page 1814
$\mathbf{DL}_{Prog}$:	Tool length delta value **DL** from the tool call or the compensation table The most recently programmed value takes effect. **Further information:** "Tool call by TOOL CALL", Page 277 **Further information:** "Tool compensation with compensation tables", Page 1044

NOTICE

Danger of collision!

The control uses the defined tool length from the tool table for compensating the tool length. Incorrect tool lengths will result in an incorrect tool length compensation. The control does not perform tool length compensation or a collision check for tools with a length of **0** and after **TOOL CALL 0**. Danger of collision during subsequent tool positioning movements!

- Always define the actual tool length of a tool (not just the difference)
- Use **TOOL CALL 0** only to empty the spindle

Tool radius compensation

The control takes the tool radius compensation into account in the following cases:

- In case of active radius compensation **RR** or **RL**
  Further information: "Tool radius compensation", Page 1038
- Within machining cycles
  Further information: "Machining Cycles", Page 419
- In case of straight lines **LN** with surface normal vectors
  Further information: "Straight line LN", Page 1050

In tool radius compensation, the control takes the delta values from the tool table and the NC program into account.

Active tool radius = **R** + $\mathbf{DR}_{TAB}$ + $\mathbf{DR}_{Prog}$

R:	Tool radius **R** from the tool table **Further information:** "Tool table tool.t", Page 1814
$\mathbf{DR}_{TAB}$:	Tool radius delta value **DR** from the tool table
$\mathbf{DR}_{Prog}$:	Tool radius delta value **DR** from the tool call or the compensation table The most recently programmed value takes effect. **Further information:** "Tool call by TOOL CALL", Page 277 **Further information:** "Tool compensation with compensation tables", Page 1044

Tool data within variables

When executing a tool call, the control calculates all tool-specific values and saves them within variables.

Further information: "Preassigned Q parameters", Page 1273

Active tool length and tool radius:

Q parameters	Function
Q108	ACTIVE TOOL RADIUS
Q114	ACTIVE TOOL LENGTH

After the control has saved the current values within variables, the variables can be used in the NC program.

Application example

Use Q parameter **Q108 ACTIVE TOOL RADIUS** and the tool length delta values to shift the tool location point of a ball-nose cutter to the ball center.

```
11 TOOL CALL "BALL_MILL_D4" Z S10000
12 TOOL CALL  DL-Q108
```

This allows the control to monitor the complete tool for collisions and the dimensions used in the NC program can still be programmed with reference to the ball center.

Notes

- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.

  Further information: "Simulation of tools", Page 1414
- The machine manufacturer uses the optional machine parameter **progToolCallDL** (no. 124501) to define whether the control will consider delta values from a tool call in the **Positions** workspace.

  Further information: "Tool call", Page 277

  Further information: "Positions workspace", Page 141
- The control takes up to six axes including the rotary axes into account in the tool compensation.

16.2 Tool radius compensation

Application

When tool radius compensation is active, the control will no longer reference the positions in the NC program to the tool center point, but to the cutting edge.

Use tool radius compensation to program drawing dimensions without having to consider the tool radius. This lets you use a tool with deviating dimensions without having to modify the program after a tool has broken.

Related topics

- Presets on the tool

  Further information: "Presets on the tool", Page 244

Requirements

- Defined tool data in the tool management

  Further information: "Tool management ", Page 270

Description of function

The control takes the active tool radius into account during tool radius compensation. The active tool radius results from the tool radius R and the delta values **DR** from the tool management and the **NC program**.

Active tool radius = $\mathbf{R} + \mathbf{DR}_{TAB} + \mathbf{DR}_{Prog}$

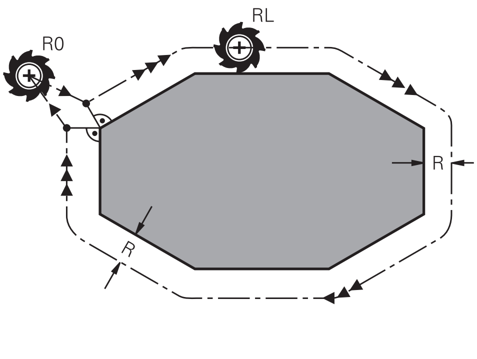
Further information: "Tool compensation for tool length and radius", Page 1036

Paraxial traverses can be compensated as follows:

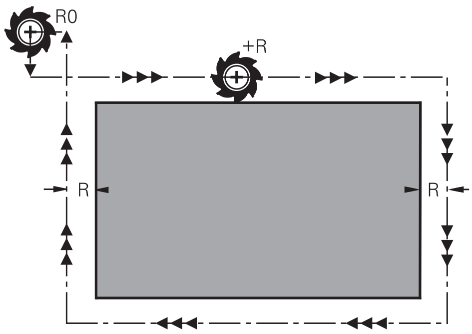
- **R+**: lengthens a paraxial traverse by the amount of the tool radius
- **R-**: shortens a paraxial traverse by the amount of the tool radius

An NC block with path functions can contain the following types of tool radius compensation:

- **RL**: tool radius compensation, on the left of the contour
- **RR**: tool radius compensation, on the right of the contour
- **R0**: resets an active tool radius compensation, positioning with the tool center point

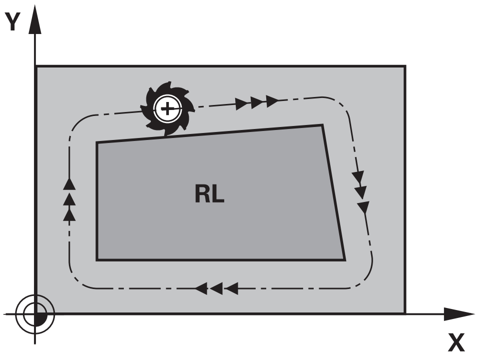


Radius-compensated traverse with path functions

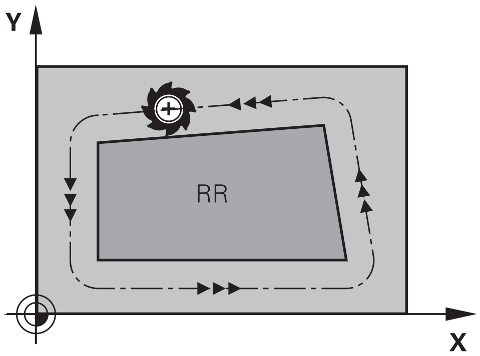


Radius-compensated traverse with paraxial movements

The tool center moves along the contour at a distance equal to the radius. **Right** or **left** are to be understood as based on the direction of tool movement along the workpiece contour.



RL: The tool moves on the left of the contour



RR: The tool moves on the right of the contour

Effect

Tool radius compensation is effective starting from the NC block in which tool radius compensation is programmed. Tool radius compensation is effective modally and at the end of the block.

> Program tool radius compensation only once, allowing e. g. changes to be implemented quicker.

The control resets tool radius compensation in the following cases:

- Positioning block with **R0**
- **DEP** function for departing from the contour
- Selection of a new NC program

Notes

NOTICE

Danger of collision!

The control needs safe positions for contour approach and departure. These positions must enable the control to perform compensating movements when radius compensation is activated and deactivated. Incorrect positions can lead to contour damage. Danger of collision during machining!

- Program safe approach and departure positions at a sufficient distance from the contour
- Consider the tool radius
- Consider the approach strategy

- When tool radius compensation is active, the control displays an icon in the **Positions** workspace.

  Further information: "Positions workspace", Page 141
- Between two NC blocks with different radius compensations **RR** and **RL** you must program at least one traversing block in the working plane without radius compensation (that is, with **R0**).
- The control takes up to six axes including the rotary axes into account in the tool compensation.

Notes in connection with the machining of corners

- Outside corners:
  If you program radius compensation, the control moves the tool around outside corners on a transitional arc. If necessary, the control reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction
- Inside corners:
  The control calculates the intersection of the tool center paths at inside corners under radius compensation. From this point it then starts the next contour element. This prevents damage to the workpiece at the inside corners. The permissible tool radius, therefore, is limited by the geometry of the programmed contour.

16.3 Tooth radius compensation for turning tools (option 50)

Application

The tip of a lathe tool has a certain radius (**RS**). When machining tapers, chamfers and radii, this results in distortions on the contour because the programmed traverse paths refer to the theoretical tool tip S. TRC prevents the resulting deviations.

Related topics

- Tool data of turning tools

  Further information: "Tool data", Page 249
- Radius compensation with **RR** and **RL** in milling mode

  Further information: "Tool radius compensation", Page 1038

Requirement

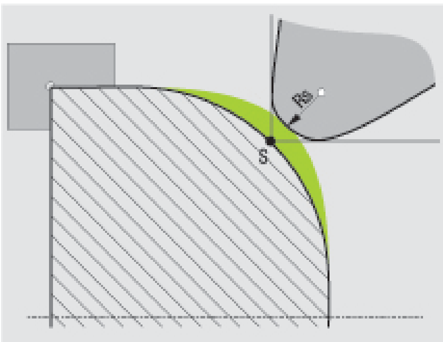
- Combined milling/turning (software option 50)
- Required tool data defined for the tool type

  Further information: "Tool data for the tool types", Page 257

Description of function

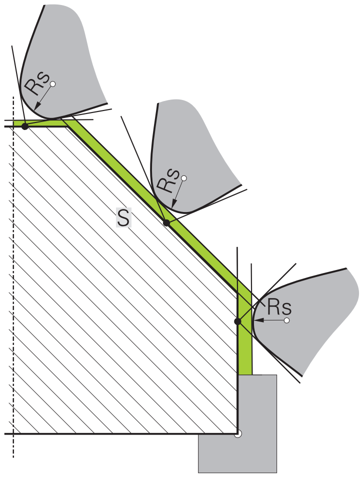
The control checks the cutting geometry with the point angle **P-ANGLE** and the setting angle **T-ANGLE**. Contour elements in the cycle are processed by the control only as far as this is possible with the specific tool.

In the turning cycles the control automatically carries out tool radius compensation. In specific traversing blocks and within programmed contours, activate TRC with **RL** or **RR**.



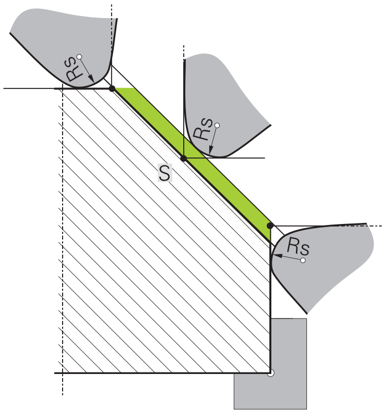
Offset between the tooth radius **RS** and the theoretical tool tip S.

Theoretical and virtual tool tip



Inclined surface with theoretical tool tip

The theoretical tool tip is effective in the tool coordinate system. When the tool is inclined, the position of the tool tip rotates with the tool.



Inclined surface with virtual tool tip

Use **FUNCTION TCPM** with the selection **REFPNT TIP-CENTER** to activate the virtual tool tip. Correct tool data are the prerequisite for calculating the virtual tool tip.

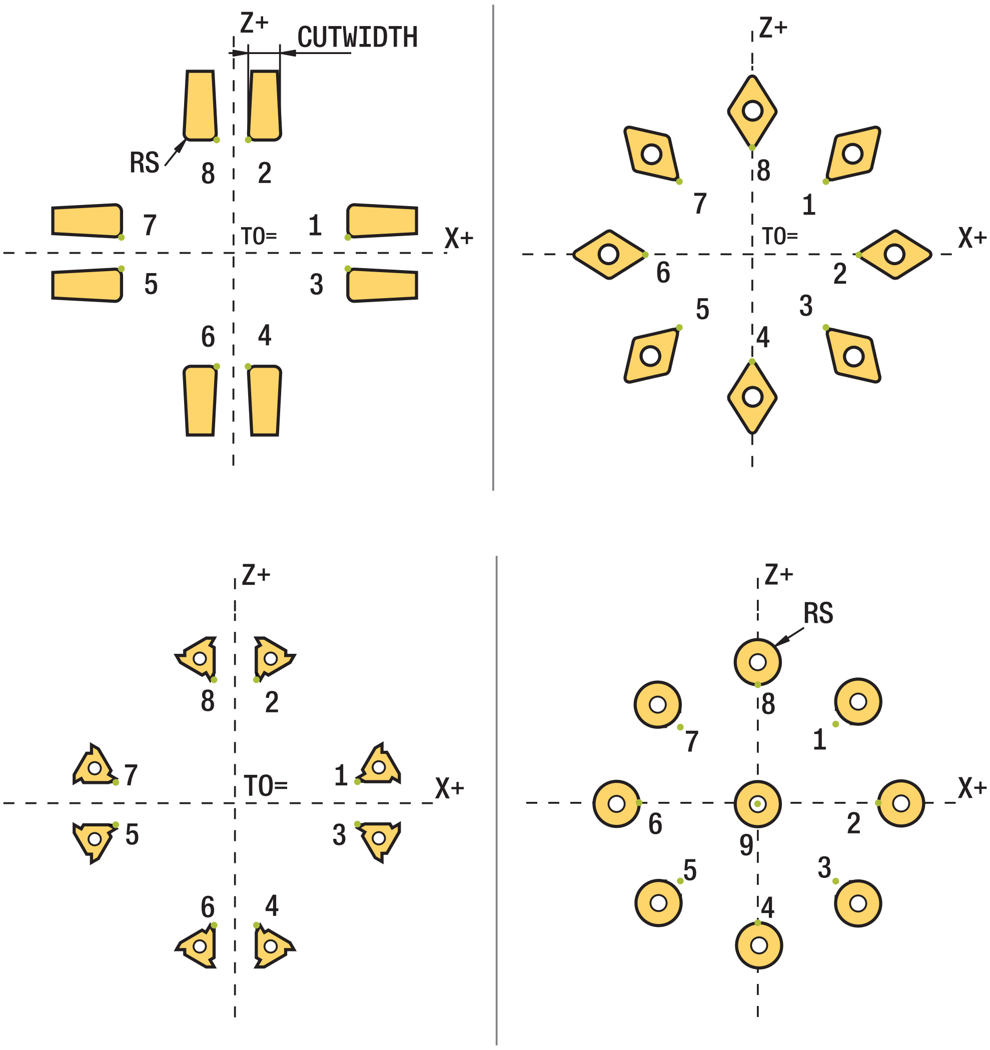
Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027

The virtual tool tip is effective in the workpiece coordinate system. When the tool is inclined, the virtual tool tip remains unchanged as long as the tool orientation **TO** is the same. The control automatically switches the status display **TO** and thus also the virtual tool tip if the tool leaves the angle range valid for **TO 1**, for example.

The virtual tool tip enables you to perform inclined paraxial longitudinal and transverse machining operations with high contour accuracy even without radius compensation.

Further information: "Simultaneous turning", Page 218

Notes



- The direction of the radius compensation is not clear when the tool-tip position (**TO=2, 4, 6, 8**) is neutral. In this case, TRC is only possible within fixed machining cycles.
- Tool tip radius compensation is also possible during inclined machining.

  Active miscellaneous functions limit the possibilities here:
  - With **M128** tool-tip radius compensation is possible only in combination with machining cycles
  - **M144** or **FUNCTION TCPM** with **REFPNT TIP-CENTER** also allows tool-tip radius compensation with all positioning blocks, e.g. with **RL**/**RR**
- The control displays a warning when residual material is left behind due to the angle of the secondary cutting edges. You can suppress the warning with the machine parameter **suppressResMatlWar** (no. 201010).

16.4 Tool compensation with compensation tables

Application

With the compensation table, you can save compensations in the tool coordinate system (T-CS) or in the working plane coordinate system (WPL-CS). The saved compensations can be called during the NC program for compensating the tool.

The compensation tables offer the following benefits:

- Values can be changed without adapting the NC program
- Values can be changed during NC program run

Via the file name extension, you can determine in which coordinate system the control will perform the compensation.

The control provides the following compensation tables:

- tco (tool correction): Compensation in the tool coordinate system (**T-CS**)
- wco (workpiece correction): Compensation in the working plane coordinate system (**WPL-CS**)

Further information: "Reference systems", Page 934

Related topics

- Contents of the compensation tables

  Further information: "Compensation table *.tco", Page 1868

  Further information: "Compensation table *.wco", Page 1870
- Editing compensation tables during program run

  Further information: "Compensation during program run", Page 1795

Description of function

In order to compensate tools by using the compensation tables, the steps below are needed:

- Creating a compensation table

  Further information: "Creating a compensation table", Page 1871
- Activating the compensation table in the NC program

  Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 1046
- As an alternative, activating the compensation table manually for the program run

  Further information: "Activating the compensation tables manually", Page 1045
- Activating a compensation value

  Further information: "Activating a compensation value with FUNCTION CORRDATA", Page 1046

The compensation table values can be edited within the NC program.

Further information: "Accessing table values ", Page 1809

The values in the compensation tables can be edited even while the program is running.

Further information: "Compensation during program run", Page 1795

Tool compensation in the tool coordinate system T-CS:

The compensation table ***.tco** defines compensation values for the tool in tool coordinate system **T-CS**.

Further information: "Tool coordinate system T-CS", Page 946

The compensations have the following effects:

- In the case of milling cutters, as an alternative to the delta values in the **TOOL CALL**

  Further information: "Tool call by TOOL CALL", Page 277
- In the case of turning tools, as an alternative to **FUNCTION TURNDATA CORR-TCS** (option 50)

  Further information: "Compensating turning tools with FUNCTION TURNDATA CORR (option 50)", Page 1047
- In the case of grinding tools, as compensation for **LO** and **R-OVR** (option 156)

  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 1828

Tool compensation in the working plane coordinate system WPL-CS:

The values from the compensation tables with the ***.wco** file name extension are applied as shifts in the working plane coordinate system (**WPL-CS**).

Further information: "Working plane coordinate system WPL-CS", Page 942

The ***.wco** compensation tables are used mainly for turning (option 50).

The compensations have the following effects:

- For turning operations, as an alternative to **FUNCTION TURNDATA CORR-WPL** (option 50)
- An X shift affects the radius

The following options are available for a shift in the WPL-CS:

- **FUNCTION TURNDATA CORR-WPL**
- **FUNCTION CORRDATA WPL**
- Shifting with the turning-tool table
  - Optional **WPL-DX-DIAM** column
  - Optional **WPL-DZ** column

> The shifts programmed with **FUNCTION TURNDATA CORR-WPL** and **FUNCTION CORRDATA WPL** are alternative programming options for the same shift.
>
> A shift in the working plane coordinate system (**WPL-CS**) defined by the turning-tool table is added to the **FUNCTION TURNDATA CORR-WPL** and **FUNCTION CORRDATA WPL** functions.

Activating the compensation tables manually

The compensation tables can be activated manually for the **Program Run** operating mode.

In the **Program Run** operating mode, the **Program settings** window contains the **Tables** area. In this area, a datum table and both compensation tables can be selected in one selection window for running the program.

When activating a table, the control will highlight this table with the status **M**.

16.4.1 Selecting a compensation table with SEL CORR-TABLE

Application

If you are using compensation tables, then use the function **SEL CORR-TABLE** to activate the desired compensation table from within the NC program.

Related topics

- Activating the compensation values in the table
  Further information: "Activating a compensation value with FUNCTION CORRDATA", Page 1046
- Contents of the compensation tables
  Further information: "Compensation table *.tco", Page 1868
  Further information: "Compensation table *.wco", Page 1870

Description of function

For the NC program, both a table ***.tco** and a table ***.wco** can be selected.

Input

11 SEL CORR-TABLE TCS "TNC:\table \corr.tco"	; Select compensation table **corr.tco**

The NC function includes the following syntax elements:

Syntax element	Meaning
SEL CORR-TABLE	Syntax initiator for selecting a compensation table
TCS or **WPL**	Compensation in the tool coordinate system **T-CS** or in the working plane coordinate system **WPL-CS**
" " or **QS**	Path of table Fixed or variable name Selection by means of a selection window

16.4.2 Activating a compensation value with FUNCTION CORRDATA

Application

The **FUNCTION CORRDATA** function allows activating a row of the compensation table for the active tool.

Related topics

- Selecting a compensation table
  Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 1046
- Contents of the compensation tables
  Further information: "Compensation table *.tco", Page 1868
  Further information: "Compensation table *.wco", Page 1870

Description of function

The activated compensation values are effective up to the next tool change or until the end of the NC program.

If you change a value, then this change does not become active until the compensation is called again.

Input

11 FUNCTION CORRDATA TCS #1	; Activate row 1 of compensation table ***.tco**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION CORRDATA	Syntax initiator for activating a compensation value
TCS, **WPL** or **RESET**	Compensation in the tool coordinate system**T-CS** or in the working plane coordinate system **WPL-CS** or reset compensation
#, **" "** or **QS**	Desired table row Fixed or variable number or name Selection by means of a selection window Only when **TCS** or **WPL** are selected
TCS or **WPL**	Reset the compensation in **T-CS** or in **WPL-CS** Only if **RESET** has been selected

16.5 Compensating turning tools with FUNCTION TURNDATA CORR (option 50)

Application

With **FUNCTION TURNDATA CORR** you can define additional compensation values for the active tool. In the **TURNDATA CORR FUNCTION** you can enter delta values for tool lengths in the X direction **DXL** and in the Z direction **DZL**. The compensation values have an additive effect on the compensation values from the turning tool table.

The compensation can be defined in the tool coordinate system **T-CS** or in the working plane coordinate system **WPL-CS**.

Further information: "Reference systems", Page 934

Related topics

- Delta values in the turning tool table

  Further information: "Turning tool table toolturn.trn (option 50)", Page 1823
- Tool compensation with compensation tables

  Further information: "Tool compensation with compensation tables", Page 1044

Requirement

- Combined milling/turning (software option 50)
- Required tool data defined for the tool type

  Further information: "Tool data for the tool types", Page 257

Description of function

The coordinate system in which the compensation is effective can be defined:

- **FUNCTION TURNDATA CORR-TCS**: The tool compensation is effective in the tool coordinate system
- **FUNCTION TURNDATA CORR-WPL**: The tool compensation is effective in the workpiece coordinate system

With **FUNCTION TURNDATA CORR-TCS** you can define a cutter radius oversize **DRS**. This enables you to program an equidistant contour oversize. **DCW** allows you to compensate the recessing width of a recessing tool.

Tool compensation **FUNCTION TURNDATA CORR-TCS** is always effective in the tool coordinate system, even during inclined machining.

FUNCTION TURNDATA CORR is always effective for the active tool. A renewed **TOOL CALL** deactivates compensation again. When you exit the NC program (e.g. with PGM MGT), the control automatically resets the compensation values.

Input

11 FUNCTION TURNDATA CORR-TCS:Z/X DZL:0.1 DXL:0.05 DCW:0.1	; Tool compensation in Z direction, X direction and for the width of the recessing tool

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION TURNDATA CORR	Syntax initiator for tool compensation of a turning tool
CORR-TCS:Z/X or **CORR-WPL:Z/X**	Tool compensation in the tool coordinate system **T-CS** or in the working plane coordinate system **WPL-CS**
DZL:	Delta value for the tool length in Z direction Optional syntax element
DXL:	Delta value for the tool length in X direction Optional syntax element
DCW:	Delta value for the recessing tool width Only if **CORR-TCS:Z/X** was selected Optional syntax element
DRS:	Delta value for the cutter radius Only if **CORR-TCS:Z/X** was selected Optional syntax element

Note

During interpolation turning the functions **FUNCTION TURNDATA CORR** and **FUNCTION TURNDATA CORR-TCS** do not have any effect.

If you want to compensate a turning tool in Cycle **292 CONTOUR.TURNG.INTRP.**, compensation needs to be performed in the cycle or in the tool table.

Further information: "Cycle 292 CONTOUR.TURNG.INTRP. (option 96)", Page 628

16.6 3D tool compensation (option 9)

16.6.1 Fundamentals

The control allows 3D tool compensation in CAM-generated NC programs with surface-normal vectors.

Further information: "Straight line LN", Page 1050

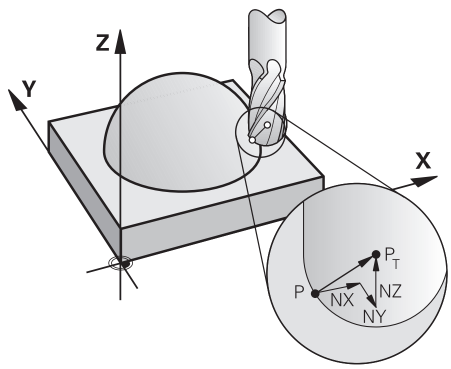
The control displaces the tool in the direction of the surface normals by the total of the delta values from tool management, tool call and compensation tables.

Further information: "Tools for 3D tool compensation", Page 1051

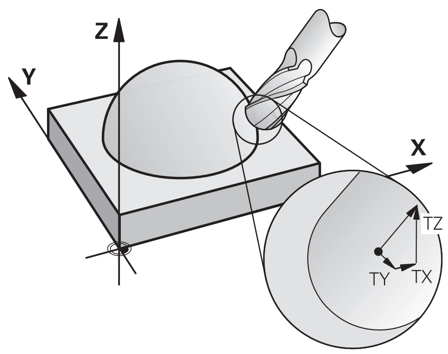
3D tool compensation can be used e. g. in the cases below:

- Compensation for re-worked tools for compensating small differences between the programmed and the actual tool dimensions
- Compensation for substitute tools with deviating diameters for compensating even larger differences between the programmed and the actual tool dimensions
- Generating a constant workpiece oversize which may serve e. g. as a finishing allowance

3D tool compensation saves time since there is no need to recalculate and output from the CAM system.



For an optional tool angle of inclination, the NC blocks must include an additional tool vector with the components TX, TY and TZ.



Note the differences between face milling and peripheral milling.

Further information: "3D tool compensation during face milling (option 9)", Page 1053

Further information: "3D tool compensation during peripheral milling (option 9)", Page 1058

16.6.2 Straight line LN

Application

Straight lines **LN** are a prerequisite for 3D compensation. Within straight lines **LN**, a surface normal vector defines the direction of the 3D tool compensation. An optional tool vector defines the tool angle of inclination.

Related topics

- Fundamentals of 3D compensation

  Further information: "Fundamentals", Page 1049

Requirements

- Advanced Functions Set 2 (software option 9)
- NC program created with a CAM system

  Straight lines **LN** cannot be programmed directly on the control, but require a CAM system.

  Further information: "CAM-generated NC programs", Page 1208

Description of function

As with a straight line **L**, a straight line **LN** is used to define the target point coordinates.

Further information: "Straight line L", Page 297

In addition, the straight lines **LN** contain a surface normal vector as well as an optional tool vector.

Input

```
LN  X+31.737 Y+21.954 Z+33.165  NX+0.2637581 NY+0.0078922 NZ-0.8764339 TX
    +0.0078922 TY-0.8764339 TZ+0.2590319 F1000 M128
```

The NC function includes the following syntax elements:

Syntax element	Meaning
LN	Syntax initiator for straight line with vectors
X, **Y**, **Z**	Coordinates of the straight-line end point
NX, **NY**, **NZ**	Components of the surface normal vector
TX, **TY**, **TZ**	Components of the tool vector Optional syntax element
R0, **RL** or **RR**	Tool radius compensation **Further information:** "Tool radius compensation", Page 1038 Optional syntax element
F, **FMAX**, **FZ**, **FU** or **F AUTO**	Feed rate **Further information:** "Cutting data", Page 280 Optional syntax element
M	Additional function Optional syntax element

Notes

- In the NC syntax, the order must be X,Y, Z for the position and NX, NY, NZ as well as TX, TY, TZ for the vectors.
- The NC syntax of LN blocks must always indicate all of the coordinates and all of the surface-normal vectors, even if the values have not changed from the previous NC block.
- Calculate the vectors as exactly as possible and specify them with at least 7 decimal places in order to avoid drastic feed rate decreases during machining.
- The CAM-generated NC program must contain normalized vectors.
- The 3D tool compensation using surface normal vectors is effective for the coordinate data specified for the main axes X, Y, Z.

Definition

Normalized vector
A normalized vector is a mathematical quantity possessing a magnitude of 1 and a direction. The direction is defined by the components X, Y and Z.

16.6.3 Tools for 3D tool compensation

Application

3D tool compensation can be used with the following tool shapes: end mill, toroid cutter and ball-nose cutter.

Related topics

- Compensation in tool management
  Further information: "Tool compensation for tool length and radius", Page 1036
- Compensation in tool call
  Further information: "Tool call by TOOL CALL", Page 277
- Compensation with compensation tables
  Further information: "Tool compensation with compensation tables", Page 1044

Description of function

The tool shapes can be distinguished by columns **R** and **R2** of the tool management:

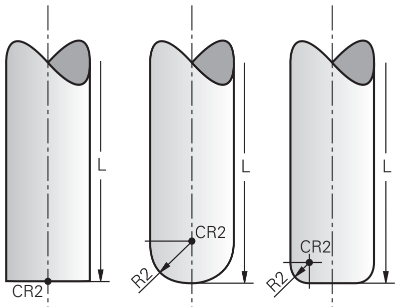
- End mill: **R2** = 0
- Toroid cutter: **R2** > 0
- Ball-nose cutter: **R2** = **R**

Further information: "Tool table tool.t", Page 1814

The delta values **DL**, **DR** and **DR2** are used to adapt the tool management values to the actual tool.

The control then compensates for the tool position by the sum of the delta values from the tool table and the programmed tool compensation (tool call or compensation table).

The surface normal vector of straight lines **LN** defines the direction in which the control compensates the tool. The surface normal vector always points to the tool radius 2 center CR2.



Position of CR2 with the individual tool shapes

Further information: "Presets on the tool", Page 244

Notes

- The tools are defined in the tool management. The overall tool length equals the distance between the tool carrier reference point and the tool tip. The control monitors the complete tool for collisions only by using the overall length.

  When defining a ball-nose cutter by the overall length and outputting an NC program to the ball center, the control must take the difference into account. When calling the tool in the NC program, define the sphere radius as a negative delta value in **DL** and thus shift the tool location point to the tool center point.
- If you load a tool with oversize (positive delta value), the control generates an error message. You can suppress the error message with the **M107** function.

  Further information: "Permitting positive tool oversizes with M107 (option 9)", Page 1262

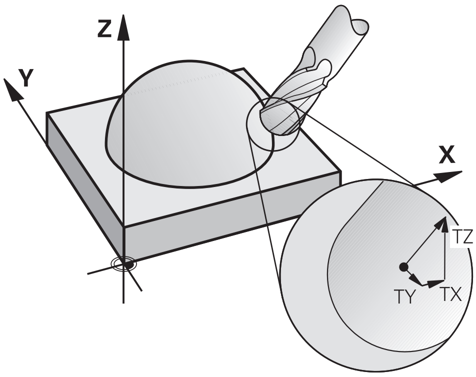
  Use the simulation to ensure that no contours are damaged by the tool oversize.

16.6.4 3D tool compensation during face milling (option 9)

Application

Face milling is a machining operation carried out with the front face of the tool.

The control displaces the tool in the direction of the surface normals by the total of the delta values from tool management, tool call and compensation tables.



Requirements

- Advanced Functions Set 2 (software option 9)
- Machine with automatically positionable rotary axes
- Output of surface normal vectors from the CAM system

  Further information: "Straight line LN", Page 1050
- NC program with **M128** or **FUNCTION TCPM**

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1245

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027

Description of function

The variants below are possible with face milling:

- **LN** block without tool orientation, **M128** or **FUNCTION TCPM** is active: Tool perpendicular to the workpiece contour
- **LN** block with tool orientation **T**, **M128** or **FUNCTION TCPM** is active: Tool keeps the set tool orientation
- **LN** block without **M128** or **FUNCTION TCPM**: The control ignores the direction vector **T** even if it is defined

Example

11 L X+36.0084 Y+6.177 Z-1.9209 R0	; No compensation is possible
12 LN X+36.0084 Y+6.177 Z-1.9209 NX-0.4658107 NY+0 NZ+0.8848844 R0	; Compensation perpendicular to the contour is possible
13 LN X+36.0084 Y+6.177 Z-1.9209 NX-0.4658107 NY+0 NZ+0.8848844 TX +0.0000000 TY+0.6558846 TZ+0.7548612 R0 M128	; Compensation is possible, DL is effective along the T vector and DR2 along the N vector
14 LN X+36.0084 Y+6.177 Z-1.9209 NX-0.4658107 NY+0 NZ+0.8848844 R0 M128	; Compensation perpendicular to the contour is possible

Notes

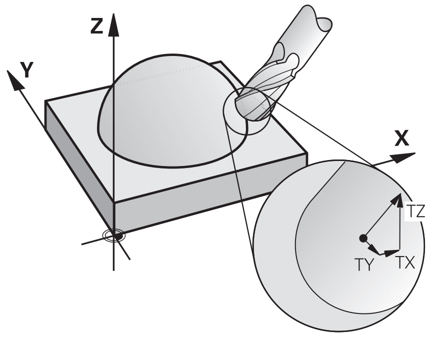
NOTICE

Danger of collision!

The rotary axes of a machine may have limited ranges of traverse, e. g. between -90° and +10° for the B head axis. Changing the tilt angle to a value of more than +10° may result in a 180° rotation of the table axis. There is a danger of collision during the tilting movement!

- Program a safe tool position before the tilting movement, if necessary.
- Carefully test the NC program or program section in the **Single Block** mode

- If no tool orientation was defined in the **LN** block and **TCPM** is active, the control maintains the tool perpendicular to the workpiece contour.

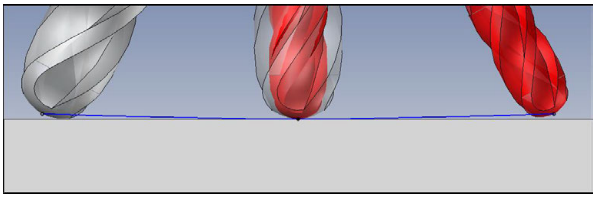


- If a tool orientation **T** has been defined in the **LN** block and M128 (or **FUNCTION TCPM**) is active at the same time, then the control will position the rotary axes automatically in such a way that the tool can reach the specified tool orientation. If you have not activated **M128** (or **TCPM FUNCTION**), then the control ignores the direction vector **T,** even if it is defined in the **LN** block.
- The control is not able to automatically position the rotary axes on all machines.
- The control generally uses the defined **delta values** for 3D tool compensation. The entire tool radius **R** + **DR**) is only taken into account if you have activated the **FUNCTION PROG PATH IS CONTOUR** function.

  Further information: "3D tool compensation with the entire tool radius with FUNCTION PROG PATH (option 9)", Page 1060

Examples

Compensate re-worked ball-nose cutter
CAM output at tool tip



Use a re-worked Ø 5.8 mm ball-nose cutter instead of Ø 6 mm.

The NC program has the following structure:

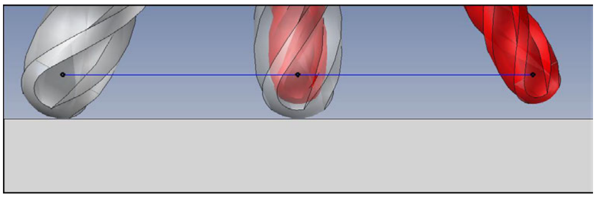
- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the tool tip
- Vector program with surface normal vectors

Proposed solution:

- Tool measurement on tool tip
- Enter the tool compensation into the tool table:
  - **R** and **R2** the theoretical tool data as from the CAM system
  - **DR** and **DR2** the difference between the nominal value and actual value

	R	R2	DL	DR	DR2
CAM	+3	+3			
Tool table	+3	+3	+0	-0.1	-0.1

Compensate re-worked ball-nose cutter
CAM output at the center of the ball



Use a re-worked Ø 5.8 mm ball-nose cutter instead of Ø 6 mm.

The NC program has the following structure:

- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the center of the sphere
- Vector program with surface normal vectors

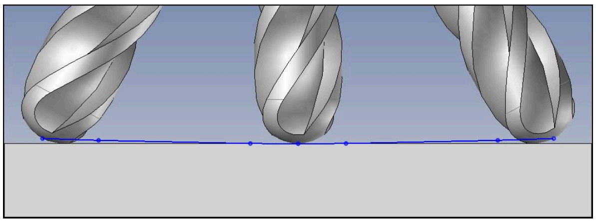
Suggested solution:

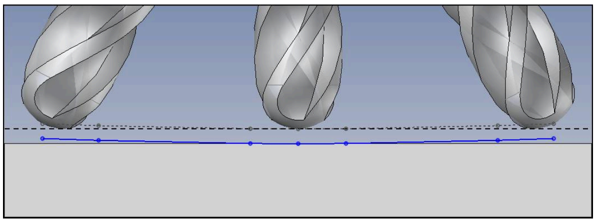
- Tool measurement on tool tip
- TCPM function **REFPNT CNT-CNT**
- Enter the tool compensation into the tool table:
  - **R** and **R2** the theoretical tool data as from the CAM system
  - **DR** and **DR2** the difference between the nominal value and actual value

	R	R2	DL	DR	DR2
CAM	+3	+3			
Tool table	+3	+3	+0	-0.1	-0.1

With TCPM **REFPNT CNT-CNT** the tool compensation values are identical for the outputs on the tool tip or center of the sphere.

Create workpiece oversize
CAM output at tool tip





Use a Ø 6 mm ball-nose cutter for achieving an even oversize of 0.2 mm on the contour.

The NC program has the following structure:

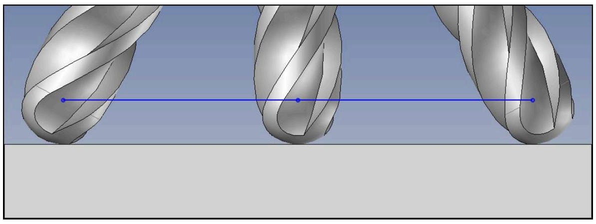
- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the tool tip
- Vector program with surface normal vectors and tool vectors

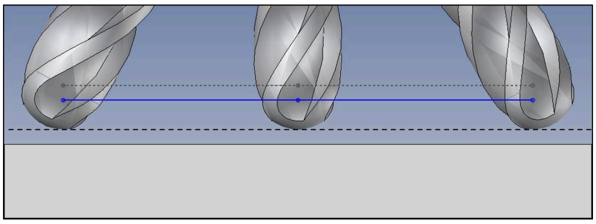
Proposed solution:

- Tool measurement on tool tip
- Enter the tool compensation into the TOOL CALL block:
  - **DL**, **DR** and **DR2** the desired oversize
- Suppress the error message with **M107**

	R	**R2**	**DL**	**DR**	**DR2**
CAM	+3	+3			
Tool table	+3	+3	+0	+0	+0
TOOL CALL			+0.2	+0.2	+0.2

Create workpiece oversize
CAM output at the center of the sphere





Use a Ø 6 mm ball-nose cutter for achieving an even oversize of 0.2 mm on the contour.

The NC program has the following structure:

- CAM output for Ø 6 mm ball-nose cutter
- NC points output on the center of the sphere
- TCPM function **REFPNT CNT-CNT**
- Vector program with surface normal vectors and tool vectors

Proposed solution:

- Tool measurement on tool tip
- Enter the tool compensation into the TOOL CALL block:
  - **DL**, **DR** and **DR2** the desired oversize
- Suppress the error message with **M107**

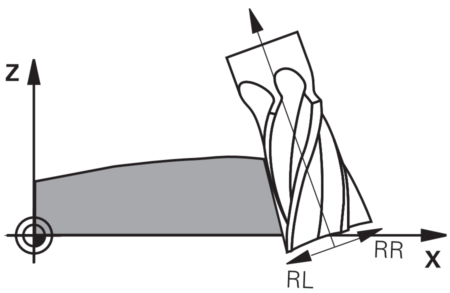
	R	R2	DL	DR	DR2
CAM	+3	+3			
Tool table	+3	+3	+0	+0	+0
TOOL CALL			+0.2	+0.2	+0.2

16.6.5 3D tool compensation during peripheral milling (option 9)

Application

Peripheral milling is a machining operation carried out with the lateral surface of the tool.

The control offsets the tool perpendicular to the direction of movement and perpendicular to the tool direction by the total of the delta values from the tool management, the tool call and the compensation tables.



Requirements

- Advanced Functions Set 2 (software option 9)
- Machine with automatically positionable rotary axes
- Output of surface normal vectors from the CAM system

  Further information: "Straight line LN", Page 1050
- NC program with spatial angles
- NC program with **M128** or **FUNCTION TCPM**

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1245

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027
- NC program with tool radius compensation **RL** or **RR**

  Further information: "Tool radius compensation", Page 1038

Description of function

The variants below are possible with peripheral milling:

- **L** block with programmed rotary axes, **M128** or **FUNCTION TCPM** is active, define compensation direction with radius compensation **RL** or **RR**
- **LN** block with tool orientation **T** perpendicular to the N vector, **M128** or **FUNCTION TCPM** is active
- **LN** block with tool orientation **T** without N vector, **M128** or **FUNCTION TCPM** is active

Example

11 L X+48.4074 Y+102.4717 Z-7.1088 C-267.9784 B-20.0115 RL M128	; Compensation is possible, compensation direction RL
12 LN X+60.6593 Y+102.4690 Z-7.1012 NX0.0000 NY0.9397 NZ0.3420 TX-0.0807 TY-0.3409 TZ0.9366 R0 M128	; Compensation is possible
13 LN X+60.6593 Y+102.4690 Z-7.1012 TX-0.0807 TY-0.3409 TZ0.9366 M128	; Compensation is possible

Notes

NOTICE

Danger of collision!

The rotary axes of a machine may have limited ranges of traverse, e. g. between -90° and +10° for the B head axis. Changing the tilt angle to a value of more than +10° may result in a 180° rotation of the table axis. There is a danger of collision during the tilting movement!

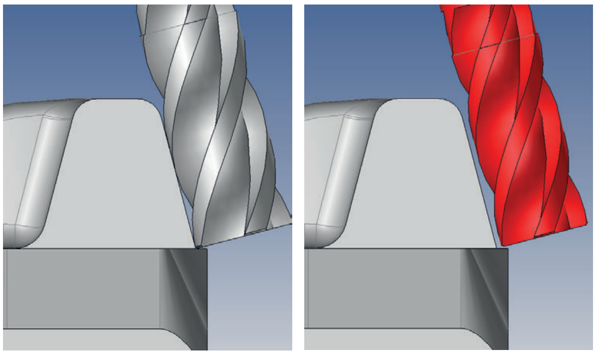
- ▶ Program a safe tool position before the tilting movement, if necessary.
- ▶ Carefully test the NC program or program section in the **Single Block** mode

- The control is not able to automatically position the rotary axes on all machines.
- The control generally uses the defined **delta values** for 3D tool compensation. The entire tool radius **R** + **DR**) is only taken into account if you have activated the **FUNCTION PROG PATH IS CONTOUR** function.

  Further information: "3D tool compensation with the entire tool radius with FUNCTION PROG PATH (option 9)", Page 1060

Example

Compensate re-worked end mill
CAM output at tool center



You use a re-worked Ø 11.8 mm end mill instead of Ø 12 mm.

The NC program has the following structure:

- CAM output for Ø 12 mm end mill
- NC points output on the tool center
- Vector program with surface normal vectors and tool vectors
  Alternative:
- Klartext program with active tool radius compensation **RL**/**RR**

Proposed solution:

- Tool measurement on tool tip
- Suppress the error message with **M107**
- Enter the tool compensation into the tool table:
  - **R** and **R2** the theoretical tool data as from the CAM system
  - **DR** and **DL** the difference between the nominal value and the actual value

	R	R2	DL	DR	DR2
CAM	+6	+0			
Tool table	+6	+0	+0	-0.1	+0

16.6.6 3D tool compensation with the entire tool radius with FUNCTION PROG PATH (option 9)

Application

The **FUNCTION PROG PATH** function defines whether the control references the 3D radius compensation only to the delta values as in the past or to the entire tool radius.

Related topics

- Fundamentals of 3D compensation
  Further information: "Fundamentals", Page 1049
- Tools for 3D compensation
  Further information: "Tools for 3D tool compensation", Page 1051

Requirements

- Advanced Functions Set 2 (software option 9)
- NC program created with a CAM system

  Straight lines **LN** cannot be programmed directly on the control, but require a CAM system.

  Further information: "CAM-generated NC programs", Page 1208

Description of function

If you activate **FUNCTION PROG PATH**, the programmed coordinates exactly correspond to the contour coordinates.

The control takes the full tool radius **R + DR** and the full corner radius **R2 + DR2** into account for 3D radius compensation.

With **FUNCTION PROG PATH OFF**, you deactivate this special interpretation.

The control only uses the delta values **DR** and **DR2** for 3D radius compensation.

If you activate **FUNCTION PROG PATH**, the interpretation of the programmed path as the contour is effective for 3D compensation movements until you deactivate the function.

Input

11 FUNCTION PROG PATH IS CONTOUR	; Use the entire tool radius for 3D compensation.

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION PROG PATH	Syntax initiator for interpreting the programmed path
IS CONTOUR or **OFF**	Use the entire tool radius or only the delta values for 3D compensation

16.7 3D radius compensation depending on the tool contact angle (option 92)

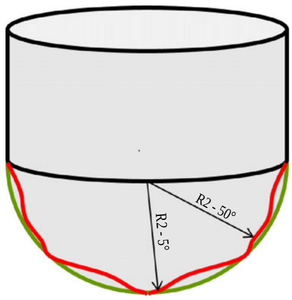
Application

Due to the production process, the effective spherical radius of a ball cutter deviates from the ideal form. The maximum form inaccuracy is defined by the tool manufacturer. Common deviations lie between 0.005 mm and 0.01 mm.

The form inaccuracy can be saved in the form of a compensation value table. This table contains angle values and the deviation from the nominal radius **R2** measured on the respective angle value.

The **3D-ToolComp** software option (option 92) enables the control to compensate the value defined in the compensation value table depending on the actual contact point of the tool.

3D calibration of the touch probe can also be carried out with the **3D-ToolComp** software option. During this process the deviations determined during touch probe calibration are saved to the compensation value table.



Related topics

- Compensation value table *.3DTC

  Further information: "*.3DTC compensation table ", Page 1871
- Touch probe 3D calibration

  Further information: "Calibrating the workpiece touch probe", Page 1439
- 3D probing with a touch probe

  Further information: "Cycle 444 PROBING IN 3-D ", Page 1668
- 3D compensation with CAM-generated NC programs with surface-normal vectors

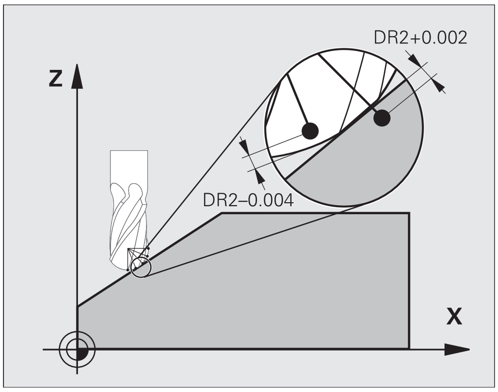
  Further information: "3D tool compensation (option 9)", Page 1049

Requirements

- Advanced Functions Set 2 (software option 9)
- 3D-ToolComp (software option 92)
- Output of surface normal vectors from the CAM system
- Tool properly defined in the tool management:
  - Value 0 in column **DR2**
  - Name of associated compensation value table in column **DR2TABLE**

  Further information: "Tool table tool.t", Page 1814

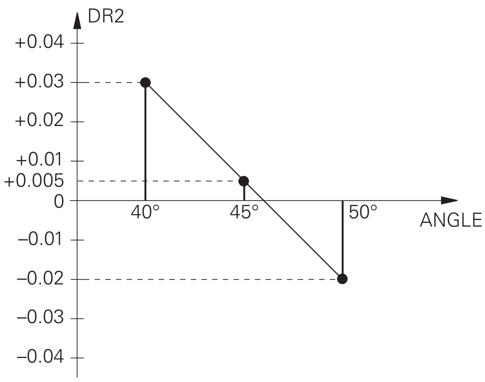
Description of function



If you are executing an NC program with surface-normal vectors and have assigned a compensation value table (DR2TABLE column) to the active tool in the tool table (TOOL.T), the control uses the values from the compensation value table instead of the compensation value DR2 from TOOL.T.

In doing so, the control takes the compensation value from the compensation value table defined for the current contact point of the tool with workpiece into account. If the contact point is between two compensation points, the control interpolates the compensation value linearly between the two closest angles.

Angle value	Compensation value
40°	0.03 mm (measured)
50°	-0.02 mm (measured)
45° (contact point)	+0.005 mm (interpolated)



Notes

- If the control cannot interpolate a compensation value, it displays an error message.
- **M107** (suppress error message for positive compensation values) is not required, even if positive compensation values are determined.
- The control uses either DR2 from TOOL.T or a compensation value from the compensation value table. Additional offsets, such as a surface oversize, can be defined via DR2 in the NC program (compensation table **.tco** or **TOOL CALL** block).

17

Files

17.1 File management

17.1.1 Basic information

Application

In the file management, the control displays drives, folders, and files. You can, for example, create or delete folders or files and can also connect drives.

The file management includes the **Files** operating mode and the **Open File** workspace.

Related topics

- Data backup
  Further information: "Backup and restore", Page 1955
- Connecting network drives
  Further information: "Network drives on the control", Page 1922

Description of function

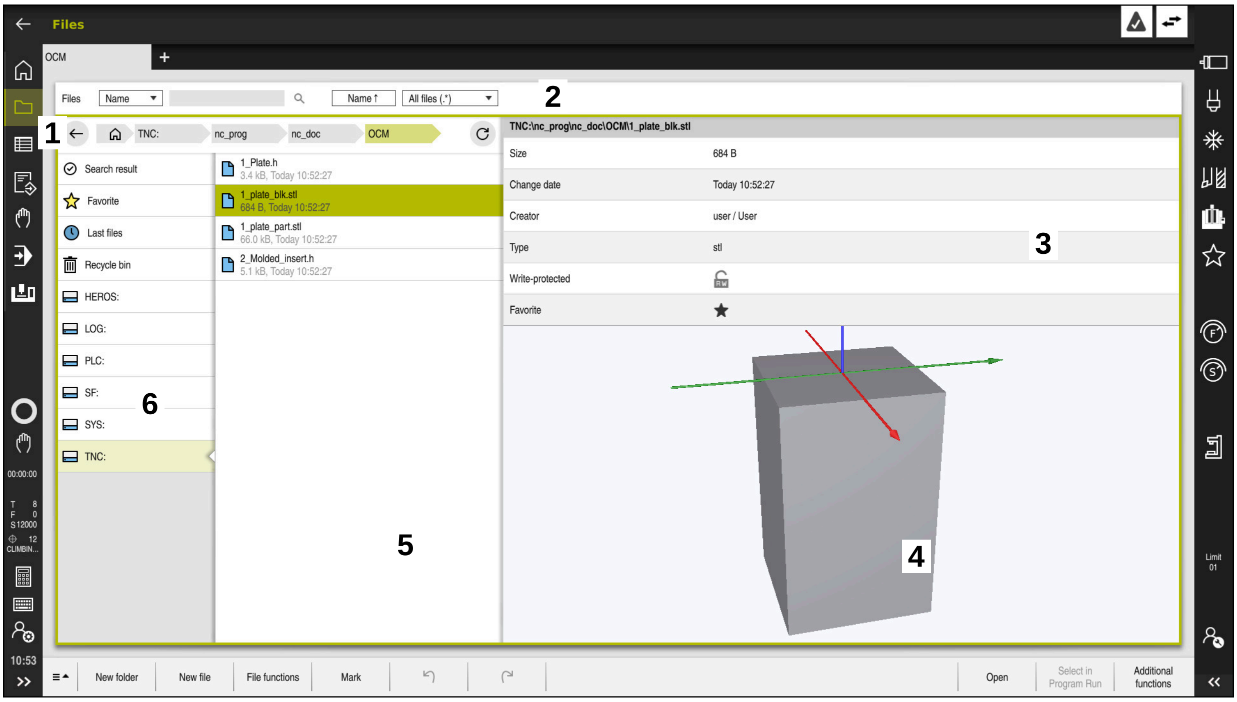
Icons and buttons

The file management contains the following icons and buttons:

Icon, button, or shortcut	Meaning
	Rename
CTRL+C	Copy
CTRL+X	Cut
	Delete
	Favorite
	Eject USB device
	Write-protection is active Only in the **Files** operating mode
	Write-protection is not active Only in the **Files** operating mode
New folder	Create new folder
New file	Create new file You create a new table in the **Tables** operating mode. **Further information:** "Tables operating mode", Page 1802

Icon, button, or shortcut	Meaning
File functions	The control opens the context menu. **Further information:** "Context menu", Page 1392 Only in the **Files** operating mode
Mark **CTRL+BLANK**	The control marks the file and opens the action bar. Only in the **Files** operating mode
CTRL+Z	Undo an action
CTRL+Y	Redo an action
Open	The control opens the file in the appropriate operating mode or application.
Select in Program Run	The control opens the file in the **Program Run** operating mode. Only in the **Files** operating mode
Additional functions	The control opens a selection menu with the following functions: ■ **Update TAB / PGM** Adapt the format and content of files from the iTNC 530 **Further information:** "Adapting an iTNC 530 file", Page 1076 ■ **Mount network share** **Further information:** "Network drives on the control", Page 1922 Only in the **Files** operating mode

Screen elements of the file management



Files operating mode

1 Navigation path

In the navigation path the control shows the position of the current folder in the folder structure. Use the individual elements of the navigation path to move to a higher folder level.

2 Title bar

- Full-text search

  Further information: "Full-text search in the title bar", Page 1069
- Sorting

  Further information: "Sorting in the title bar", Page 1069
- Filtering

  Further information: "Filtering in the title bar", Page 1069

3 Information area

Further information: "Information area", Page 1069

4 Preview area

In the preview area the control shows a preview of the selected file; for example an excerpt from an NC program.

5 Content column

In the content column the control shows all folders and files of the current drive, folder, or other source.

The control displays the following status for a file, if applicable:

- **M**: the file is active in the **Program Run** operating mode
- **S**: the file is active in the **Simulation** workspace
- **E**: the file is active in the **Editor** operating mode

6 Navigation column

Further information: "Navigation column", Page 1070

Full-text search in the title bar

Use the full-text search to look for any strings in the names or contents of files. The control searches through the active level and any lower levels of the selected drive or folder.

Use the selection menu to choose whether the control searches the names or contents of the files.

You can use the ***** character as a placeholder. This placeholder can stand for any characters or even an entire word. You can also use the placeholder to search for specific file types (e.g., ***.pdf**).

Sorting in the title bar

You can sort folders and files in ascending or descending order according to the following criteria:

- **Name**
- **Type**
- **Size**
- **Change date**

If you sort by name or type, the control lists the files alphabetically.

Filtering in the title bar

You can filter the folders and files using the following standard filters:

- **NC programs (.H, .I)**
- **Klartext file (.H)**
- **ISO file (.I)**
- **Text file (.TXT)**
- **All files (.*)**

If you want to filter for a different file type, you can use the placeholder character in the full-text search.

Further information: "Full-text search in the title bar", Page 1069

Information area

In the information area the control shows the path of the file or folder.

Further information: "Path", Page 1070

Depending on which element is selected, the control displays the following additional information:

- **Size**
- **Change date**
- **Creator**
- **Type**

You can select the following functions in the information area:

- Activate and deactivate write-protection
- Add or remove favorites

Navigation column

The navigation column offers the following possibilities for navigation:

- **Search result**

  The control displays the results of the full-text search. If there was no search, or if nothing was found, then this area is empty.
- **Favorite**

  The control displays all folders and files that you have marked as favorites.
- **Last files**

  The control displays the 15 most recently opened files.
- **Recycle bin**

  The control moves deleted folders and files to the recycle bin. You can use the context menu to restore these files or empty the recycle bin.

  Further information: "Context menu", Page 1392
- Drives (e.g., **TNC:**)

  The control displays internal and external drives (e.g., a USB device).

Permitted characters

You can use the following characters for the names of drives, folders, and files:

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _ -

Only use characters that are shown here; otherwise problems might occur (for example, during data transmission).

The following characters have specific functions, and must therefore not be used in a name:

Symbol	Function
.	Separates the file name from the file type
\ /	Separates between drive, folder, and file in the path
:	Separates the drive names

Name

When you create a file, you first define its name. The file name is followed by the file name extension, consisting of a period and the file type.

Path

The maximum permitted path length is 255 characters. The path length consists of the drive characters, the folder name, and the file name, including the file name extension.

Absolute path

An absolute path specifies the exact position of a file. The path begins with the drive and then goes through the folder structure in sequence all the way to the file (e.g., **TNC:\nc_prog\$mdi.h**). If the file being called has been moved, then a new absolute path must be entered.

Relative path

A relative path specifies the position of a file in relation to the file that is calling it. The path goes through the folder structure in sequence all the way to the file, starting from the file that is calling it (e.g., **demo\reset.H**). If a file has been moved, then a new relative path must be entered.

File types

You can use uppercase or lowercase letters to define the file type.

HEIDENHAIN-specific file types

The control can open the following HEIDENHAIN-specific file types:

File type	Application
H	NC program written in HEIDENHAIN Klartext **Further information:** "Contents of an NC program", Page 190
I	NC program with ISO commands
HC	Contour definition in the smarT.NC format of the iTNC 530
HU	Main program in the smarT.NC format of the iTNC 530
3DTC	Table with 3D tool compensations that depend on the contact angle **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1062
D	Table with workpiece datums **Further information:** "Datum table", Page 1858
DEP	Automatically generated table with data that depend on the NC program (e.g., the tool usage file) **Further information:** "Tool usage file", Page 1845
P	Table for pallet-oriented machining **Further information:** "Job list workspace", Page 1764
PNT	Table with machining positions (e.g., for the machining of irregular point patterns) **Further information:** "Point table", Page 1856
PR	Table with workpiece presets **Further information:** "Preset table", Page 1851
TAB	Freely definable table (e.g., for protocol files or as WMAT and TMAT tables for automatic calculation of cutting data) **Further information:** "Freely definable tables", Page 1850 **Further information:** "Cutting data calculator", Page 1397
TCH	Table with the assignment of the tool magazine **Further information:** "Pocket table tool_p.tch", Page 1843
T	Table with tools for all technologies **Further information:** "Tool table tool.t", Page 1814
TP	Table with touch probes **Further information:** "Touch probe table tchprobe.tp", Page 1839
TRN	Table with turning tools **Further information:** "Turning tool table toolturn.trn (option 50)", Page 1823
GRD	Table with grinding tools **Further information:** "Grinding tool table toolgrind.grd (option 156)", Page 1828

File type	Application
DRS	Table with dressing tools **Further information:** "Dressing tool table tooldress.drs (option 156)", Page 1836
TNCDRW	Contour description as a 2D drawing **Further information:** "Graphical Programming", Page 1337
M3D	Format for tool carriers or collision objects (option 40), for example **Further information:** "Options for fixture files", Page 1092
TNCBCK	File for data backup and restoration **Further information:** "Backup and restore", Page 1955

The control opens these file types with an internal application or with a HEROS tool.

Further information: "Opening files with additional software", Page 1972

Standardized file types

The control can open the following standardized file types:

File type	Application
CSV	Text file for saving or exchanging simple structured data **Further information:** "Importing and exporting tool data", Page 271
XLSX (XLS)	File type for various spreadsheet programs (e.g., Microsoft Excel)
STL	3D model created with triangular facets (e.g., fixtures) **Further information:** "Exporting a simulated workpiece as STL file", Page 1415
DXF	2D CAD files
IGS/IGES STP/STEP	3D CAD files **Further information:** "Opening CAD Files with the CAD-Viewer", Page 1355
CHM	Help files in compiled or compressed format
CFG	Configuration files of the control **Further information:** "Options for fixture files", Page 1092 **Further information:** "Machine parameters", Page 1958
CFT	3D data of a parameterizable tool-carrier template **Further information:** "Tool carrier management", Page 274
CFX	3D data of a geometrically determined tool carrier **Further information:** "Tool carrier management", Page 274
HTM/HTML	Text file with structured content of a website that can be opened in a browser (e.g., the integrated product help) **Further information:** "User's Manual as integrated product help: TNCguide", Page 62
XML	Text file with hierarchically-structured data
PDF	Document format that visually reproduces the original file identically, regardless of the source application
BAK	Data- backup file **Further information:** "Data backup", Page 1972
INI	Initialization file (e.g., can contain program settings)
A	Text file (e.g., for defining the screen output format with FN16)
TXT	Text file (e.g., for saving the results of measurement cycles with FN16)
SVG	Picture format for vector graphics
BMP GIF JPG/JPEG PNG	Picture formats for pixel graphics By default, the control uses the PNG format for screenshots **Further information:** "HEROS menu", Page 1964
OGG	Container file format for the OGA, OGV, and OGX media types
ZIP	Container file format that collects multiple compressed files.

The control opens some of these file types with the HEROS tools.

Further information: "Opening files with additional software", Page 1972

Notes

- The control has 189 GB of disk space. The maximum size of any file is limited to 2 GB.
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). These characters can cause problems when inputting or reading data in conjunction with SQL commands.

  Further information: "Table access with SQL statements", Page 1316
- If the cursor is within the content column, you can start inputting through the keyboard. The control opens a separate input field and automatically searches for the entered string. If it finds a file or folder with that string, then the control moves the cursor to it.
- If you exit an NC program by pressing the **END BLK** key, the control opens the **Add** tab. The cursor is on the NC program that was just closed.

  If you press the **END BLK** key again, the control opens the NC program again with the cursor on the last selected line. With large files, this behavior can cause a delay.

  If you press the **ENT** key, the control always opens an NC program with the cursor on line 0.
- The control creates dependency files with the ***.dep** extension for the tool-usage file e. g. in order to perform a tool usage test.

  Further information: "Tool usage test", Page 284

  In the machine parameter **dependentFiles** (no. 122101) the machine manufacturer defines whether the control displays dependency files.
- In the machine parameter **createBackup** (no. 105401) the machine manufacturer defines whether the control creates a backup file when saving an NC program. Please note that these backup files will take up disk space.

Hints about file functions

If you select a file or folder and swipe to the right, the control displays the following file functions:

- Rename
- Copy
- Cut
- Delete
- Mark as favorite

You can also select these file functions from the context menu.

Further information: "Context menu", Page 1392

Hints about copied files

- If you copy a file and then paste it to the same folder, the control adds the suffix **_Copy** to the file name.
- If you paste a file to another folder and that folder contains a file with the same name, the control opens the **Insert file** window. The control displays the path of the two files and offers the following options:
  - Replace existing file
  - Skip copied file
  - Add suffix to file name

  You can also apply the selected option to all such cases.

17.1.2 Open File workspace

Application

The **Open File** workspace allows e. g. selecting and creating files.

Functionality

The **Open File** workspace can be opened by the icons below, depending on the active operating mode:

Icon	Function
	**Add** in the **Tables** and **Editor** operating modes
	**Open File** in the **Program Run** operating mode

The functions below can be executed in the **Open File** workspace in the respective operating modes:

Function	Tables operating mode	Editor operating mode	Program Run operating mode
New folder	✓	✓	–
New file	✓	✓	–
Open	✓	✓	✓

17.1.3 Quick selection workspace

Application

In the **Quick selection** workspace you open an existing table or create a file, e. g. an NC program.

Description of function

The **Quick selection** workspace can be opened with the **Add** function in the **Tables** and **Editor** operating modes.

Further information: "Icons on the control's user interface", Page 102

In the **Tables** operating mode the tables below can be opened:

- **Tool management**
- **Pocket table**
- **Presets**
- **Touch probes**
- **Datums**
- **T usage order**
- **Tooling list**

Use the **Create new table** button to create various tables on the control.

In the **Editor** operating mode the files below can be created:

- **New program**
- **New contour**
- **New job list**

17.1.4 Adapting an iTNC 530 file

Application

In order to use a file created on the iTNC 530 on the **TNC7** as well, the control must adapt the file's format and content. The **Update TAB / PGM** function does this job.

Description of function

Importing an NC program

The control uses the **Update TAB / PGM** function to remove umlauts and checks if the NC block **END PGM** exists. The NC program would be incomplete without this NC block.

Importing a tool table

The following characters are permitted in the **NAME** column of the tool table:

$ % & , - . 0 1 2 3 4 5 6 7 8 9 @ A B C D E F G H I J K L M N O P Q R S T U V W X Y Z _

The control converts a comma into a dot with the **Update TAB / PGM** function.

The control adopts all supported tool types and assigns the **Undefined** type to all unknown tool types.

Further information: "Tool table tool.t", Page 1814

Adapting a file

Prepare a backup of the original file before adapting

To adapt the format and the content of an iTNC 530 file:

- Select the **Files** operating mode
- Select the desired file

Additional functions

- Select **Additional functions**
- > The control displays a selection menu.
- Select **Update TAB / PGM**
- > The control adapts the file format and content.

> The control saves the changes and overwrites the original file.

- Check the content after adapting

Notes

- The machine manufacturer uses import and update rules to define which adaptations the control is to execute, e. g. removal of umlauts.
- The machine manufacturer uses the optional machine parameter **importFromExternal** (no. 102909) to define for each file type if automatic adaptation is carried out upon copying to the control.

17.1.5 USB devices

Application

A USB device allows transmitting data and saving data externally.

Requirement

- USB 2.0 or 3.0
- USB device with supported file system

  The control supports USB devices with the following file systems:
  - FAT
  - VFAT
  - exFAT
  - ISO9660

  > The control does not support USB devices with other file systems, such as NTFS.

- A ready data interface

  Further information: "Serial data transfer", Page 1968

Description of function

The control displays a USB device as a drive in the navigation column of the **Files** operating mode or of the **Open File** workspace.

The control automatically detects USB devices. If you connect a USB device with a file system that is not supported, the control generates an error message.

Before executing an NC program saved on the USB device, the file must be transferred to the control hard disk.

When transmitting large files, the control displays the data transmission progress at the bottom of the navigation and content column.

Removing a USB device

To remove a USB device:

- ▶ Select **Eject**
- > The control opens a pop-up window and asks whether you want to eject the USB device.

OK

- ▶ Press **OK**
- > The control shows the message **The USB device can be removed now**.

Notes

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

- ▶ Copy the NC program and all called files to the **TNC:** drive

NOTICE

Caution: Data may be lost!

Always remove a connected USB device properly, otherwise data may be damaged or deleted!

- ▶ Use the USB port for transfer and backup only; do not use it for editing and executing NC programs
- ▶ Use the icon to remove USB devices when data transfer is complete

- If an error message is displayed when connecting a USB device, check the setting in the **SELinux** security software.
  Further information: "SELinux security software", Page 1921
- If the control displays an error message when using a USB hub, ignore and acknowledge the message with the **CE** key.
- Prepare a backup of the files on the control at regular intervals.
  Further information: "Data backup", Page 1972

17.2 Programmable file functions

Application

Programmable file functions enable management of files from within the NC program. Files can be opened, copied, relocated and deleted. This allows e. g. opening the drawing of a component during the measuring process with a touch probe cycle.

Description of function

Opening a file with OPEN FILE

The **OPEN FILE** function allows you to open a file from within an NC program.

If you define **OPEN FILE**, the control continues the dialog and you can program a **STOP**.

Using this function, the control can open all file types that you can open manually.

Further information: "File types", Page 1071

The control opens the file in the HEROS tool last used for this file type. If you have never opened a file of a certain file type and multiple HEROS tools are available, the control will interrupt program run and open the **Application?** window. In the **Application?** window, you can select the HEROS tool the control should use to open the file. The control saves this selection.

Multiple HEROS tools are available for opening the following file types:

- CFG
- SVG
- BMP
- GIF
- JPG/JPEG
- PNG

> In order to avoid program run interruptions or having to select an alternative HEROS tool, open a file of the corresponding file type once in the file manager. If the files of a certain file type can be opened in multiple HEROS tools, you can use the file manager to select the HEROS tool to be used for opening files of this file type.
>
> **Further information:** "File management", Page 1066

This function is effective in the **Simulation** workspace, the **MDI** application, and the **Program Run** operating mode.

Input

```
11 OPEN FILE "FILE1.PDF" STOP
```

The NC function includes the following syntax elements:

Syntax element	Meaning
OPEN FILE	Start of syntax for the OPEN FILE function
" "	Path of the file to be opened
STOP	Interrupts the program run or simulation Optional syntax element

Copying, relocating and deleting files with FUNCTION FILE

The control offers the functions below for copying, moving and deleting files from an NC program:

NC function	Description
FUNCTION FILE COPY	This function copies a file into a target file. The control substitutes the content of the target file. This function requires specifying the path to both files.
FUNCTION FILE MOVE	This function moves a file to a target file. The control substitutes the content of the target file and deletes the file to be moved. This function requires specifying the path to both files.
FUNCTION FILE DELETE	This function deletes the selected file. This function requires specifying the path to the file to be deleted.

These functions are effective in the **MDI** application and in the **Program Run** operating mode.

Input

```
11 FUNCTION FILE COPY "FILE1.PDF" TO "FILE2.PDF"    ; Copy the file from the NC program
```

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE COPY	Syntax initiator for the Open file function
" "	Path of the file to be copied
" "	Path of the file to be substituted

```
11 FUNCTION FILE MOVE "FILE1.PDF" TO "FILE2.PDF"    ; Move the file from the NC program
```

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE MOVE	Syntax initiator for the Move file function
" "	Path of the file to be relocated
" "	Path of the file to be substituted

```
11 FUNCTION FILE DELETE "FILE1.PDF"    ; Delete the file from the NC program
```

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE DELETE	Syntax initiator for the Delete file function
" "	Path of the file to be deleted

Notes

NOTICE

Caution: Data may be lost!

When deleting a file with the **FUNCTION FILE DELETE** function, the control will not put this file into the recycle bin. The control deletes the file once and for all!

- ▶ Use this function only with files that are no longer needed

- There are various ways to select files:
  - Enter the file path
  - Select the file in a select window
  - Define the file path or name of the subprogram in a QS parameter

    If the called file is located in the same directory as the calling file, you may also enter just the file name.
- When applying file functions relating to the calling NC program in a called NC program, the control will display an error message.
- When intending to copy or move a non-existent file, the control displays an error message.
- If the file to be deleted does not exist, the control does not display an error message.

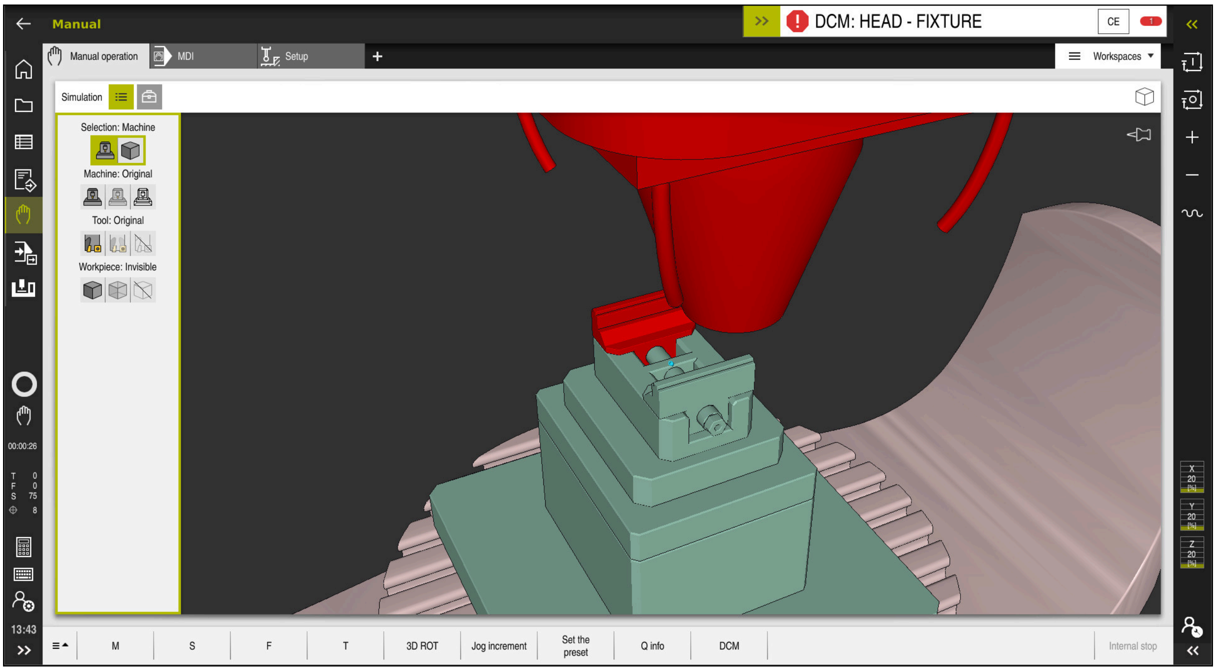
18

Collision Monitoring

18.1 Dynamic Collision Monitoring (DCM, option 40)

Application

Dynamic Collision Monitoring (DCM, dynamic collision monitoring) can be used for collision monitoring of machine components defined by the machine manufacturer. When the collision objects come closer to each other than a defined minimum distance, the control stops and displays an error message. This procedure reduces the risk of collision.



Dynamic Collision Monitoring (DCM) including collision warning

Requirements

- Dynamic Collision Monitoring (DCM, software option 40)
- Control prepared by the machine manufacturer

  The machine manufacturer must define a kinematics model of the machine, insertion point for fixtures and the safety distance between collision objects.

  Further information: "Fixture monitoring (option 40)", Page 1091
- Tools with a positive radius **R** and length **L**.

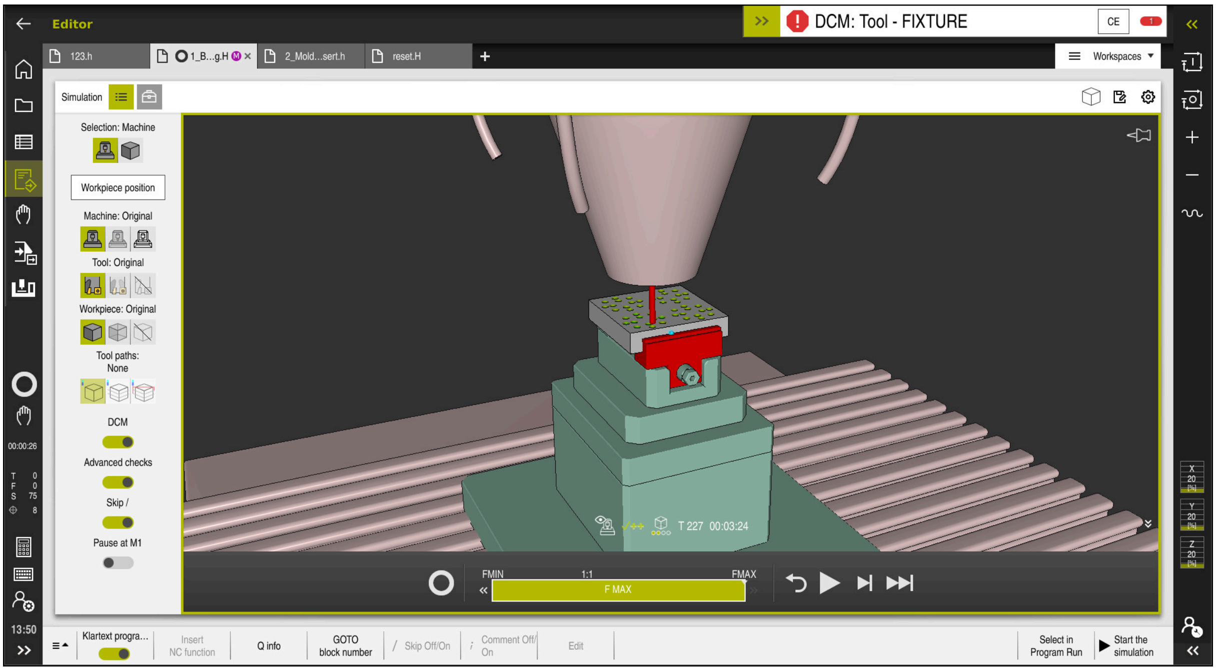
  Further information: "Tool table tool.t", Page 1814
- The values in the tool management equal the actual tool dimensions

  Further information: "Tool management ", Page 270

Description of function

Refer to your machine manual.
The machine manufacturer adapts the Dynamic Collision Monitoring (DCM) to the control.

The machine manufacturer can define machine components and minimum distances that are to be monitored by the control during all machine movements. If two collision objects come closer to each other than a defined minimum distance, the control generates an error message and terminates the movement.



Dynamic Collision Monitoring (DCM) in the **Simulation** workspace

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- Make sure to activate DCM whenever possible
- Make sure to always re-activate DCM immediately after a temporary deactivation
- Carefully test your NC program or program section in the **Single Block** mode while DCM is deactivated

The control displays the collision objects graphically in the following operating modes:

- **Editor** operating mode
- **Manual** operating mode
- **Program Run** operating mode

The control also monitors the tools, as defined in tool management, for collision.

NOTICE

Danger of collision!

Even if Dynamic Collision Monitoring (DCM) is active, the control does not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!

- Enable the **Advanced checks** switch for simulations
- Check the machining sequence using a simulation
- Carefully test your NC program or program section in the **Single Block** mode

Further information: "Advanced checks in the simulation", Page 1109

Dynamic Collision Monitoring (DCM) in the Manual and Program Run operating modes

Dynamic Collision Monitoring (DCM) is activated separately for the **Manual** and **Program Run** operating modes, using the **DCM** button.

Further information: "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 1088

In the **Manual** and **Program Run** operating modes, the control stops the movement if two collision objects approach each other by less than a minimum distance. In this case, the control displays an error message naming the two objects causing collision.

Refer to your machine manual.
The machine manufacturer can define the minimum distance between two collision-monitored objects.

Before the collision warning, the control dynamically reduces the feed rate of movements. This ensures that the axes stop in good time before a collision occurs.

When the collision warning is triggered, the control displays the colliding objects in red in the **Simulation** workspace.

When a collision warning has been issued, machine movements via the axis direction keys or the handwheel are only possible if they increase the distance between the collision objects.
With active collision monitoring and a simultaneous collision warning, no movements are permitted that reduce the distance or leave it unchanged.

Dynamic Collision Monitoring (DCM) in the Editor operating mode

Dynamic Collision Monitoring (DCM) is activated for simulation in the **Simulation** workspace.

Further information: "Activating Dynamic Collision Monitoring (DCM) for the simulation", Page 1088

In the **Editor** operating mode, an NC program can be collision-monitored even prior to execution. In case of collision, the control stops the simulation and displays an error message naming the two objects causing collision.

HEIDENHAIN recommends the use of Dynamic Collision Monitoring (DCM) in the **Editor** operating mode only in addition to DCM in the **Manual** and **Program Run** operating modes.

> The enhanced collision monitoring shows collisions between the workpiece and tools or tool holders.
>
> **Further information:** "Advanced checks in the simulation", Page 1109

To obtain a simulation result that is similar to the program run, the following aspects must match:

- Workpiece preset
- Basic rotation
- Offsets of each axis
- Tilting condition
- Active kinematic model

The active workpiece preset for the simulation must be selected. The active workpiece preset from the preset table can be adopted into the simulation.

Further information: "Visualization options column", Page 1407

In a simulation, the following aspects may differ from the actual machine or may not be available at all:

- The simulated tool change position may differ from the tool change position in the machine.
- Changes in the kinematics may have a delayed effect in the simulation.
- PLC positioning movements are not displayed in the simulation.
- Global program settings (GPS, option 44) are not available
- Handwheel superimposition is not available
- Editing of job lists is not available
- Traverse range limits from the **Settings** application are not available.

18.1.1 Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- Make sure to activate DCM whenever possible
- Make sure to always re-activate DCM immediately after a temporary deactivation
- Carefully test your NC program or program section in the **Single Block** mode while DCM is deactivated

To active Dynamic Collision Monitoring (DCM) for the **Manual** and **Program Run** operating modes:

- Select the **Manual** operating mode
- Select the **Manual** application
- Select **DCM**
- > The control opens the **Dyna. Coll. Monitoring (DCM)** window.
- Activate DCM in the desired operating modes, using the switches
- Press **OK**
- > The control activates DCM in the selected operating modes.

The control displays the status of Dynamic Collision Monitoring (DCM) in the **Positions** workspace. When deactivating DCM, the control displays an icon in the information bar.

18.1.2 Activating Dynamic Collision Monitoring (DCM) for the simulation

Dynamic Collision Monitoring (DCM) can be activated for the simulation only in the **Editor** operating mode.

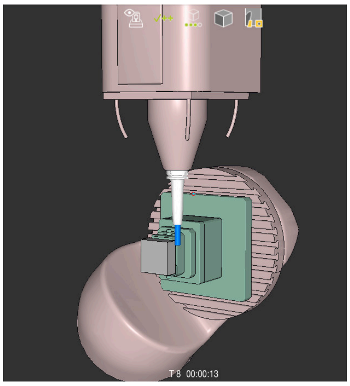
To activate DCM for the simulation:

- Select the **Editor** operating mode
- Select **Workspaces**
- Select **Simulation**
- > The control opens the **Simulation** workspace.
- Select the **Visualization options** column
- Activate the **DCM** switch
- > The control activates DCM in the **Editor** operating mode.

The control displays the status of Dynamic Collision Monitoring (DCM) in the **Simulation** workspace

Further information: "Icons in the Simulation workspace", Page 1406

18.1.3 Activating the graphic display of the collision objects



Simulation in the **Machine** mode

To activate the graphic display of the collision objects:

- Select an operating mode, e. g. **Manual**
- Select **Workspaces**
- Select the **Simulation** workspace
- > The control opens the **Simulation** workspace.
- Select the **Visualization options** column
- Select the **Machine** mode
- > The control displays a graphic representation of the machine and the workpiece.

Changing the representation

To change the graphic display of the collision objects:

- Activate the graphic display of the collision objects
- Select the **Visualization options** column
- Change the graphic display of the collision objects, e. g. **Original**

18.1.4 FUNCTION DCM: Deactivating and activating Dynamic Collision Monitoring (DCM) in NC programs

Application

Some machining steps are by design performed close to a collision object. If you want to exclude some machining steps from Dynamic Collision Monitoring (DCM), you can deactivate DCM for them in your NC program. This means that it is possible to monitor individual parts of an NC program for collision.

Requirement

This function can only be used if Dynamic Collision Monitoring (DCM) is active for the **Program Run** operating mode. Otherwise, the function has no effect and you cannot activate DCM from within the NC program.

Description of function

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- Make sure to activate DCM whenever possible
- Make sure to always re-activate DCM immediately after a temporary deactivation
- Carefully test your NC program or program section in the **Single Block** mode while DCM is deactivated

FUNCTION DCM is only effective within the NC program.

It is for example possible to deactivate Dynamic Collision Monitoring (DCM) in the following situations in your NC program:

- To reduce the distance between two objects monitored for collision
- To prevent stops during program runs

The following NC functions are available:

- **FUNCTION DCM OFF** deactivates collision monitoring until the end of the NC program or the call of the **FUNCTION DCM ON** function.
- **FUNCTION DCM ON** revokes the **FUNCTION DCM OFF** function and reactivates collision monitoring.

Programming the FUNCTION DCM function

To program the **FUNCTION DCM** function:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **FUNCTION DCM**
- Select the **OFF** or **ON** syntax element

Notes

- Dynamic Collision Monitoring (DCM) helps you reduce the risk of collision. However, the control cannot consider all possible constellations during operation.
- The control can protect only those machine components from collision that your machine manufacturer has defined correctly with regard to dimensions, orientation, and position.
- The control takes the **DL** and **DR** delta values from the tool management into account. Delta values from the **TOOL CALL** block or a compensation table are not taken into account.
- For certain tools, e. g. face-milling cutters, the radius that would cause a collision can be greater than the value defined in the tool management.
- When a touch probe cycle starts, the control no longer monitors the stylus length and ball-tip diameter, so you can still probe collision objects.

18.2 Fixture monitoring (option 40)

18.2.1 Fundamentals

Application

The Fixture Monitoring function allows you to map setup situations and monitor them for collisions.

Related topics

- Dynamic Collision Monitoring (DCM, option 40)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084
- Integrating an STL file as workpiece blank

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 239

Requirements

- Dynamic Collision Monitoring (DCM, software option 40)
- Kinematics description

  The machine manufacturer creates the kinematics description
- Insertion point defined

  Using the insertion point, the machine manufacturer defines the preset for positioning the fixtures. The insertion point is often located at the end of the kinematic chain, e. g. at the center of a rotary table. For information about the position of the insertion point, please refer to your machine manual.
- Fixtures of suitable format:
  - STL file
    - 20,000 triangles maximum
    - Triangular mesh forms a closed shell
  - CFG file
  - M3D file

Description of function

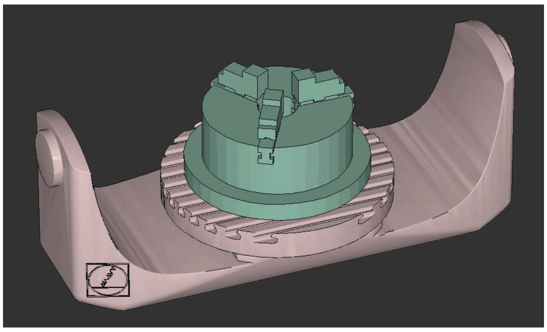
To use fixture monitoring, the steps below are needed:

- Creating a fixture or loading into the control

  Further information: "Options for fixture files", Page 1092
- Fixture placement
  - **Set up fixtures** function in the **Setup** application (option 140)

    Further information: "Integrating the fixtures into collision monitoring (option 140)", Page 1094
  - Manual fixture placement
- When changing fixtures, load or remove the fixture in the NC program

  Further information: "Loading and removing fixtures using the FIXTURE function (option 40)", Page 1102



Three-jaw chuck loaded as fixture

Options for fixture files

For the integration of fixtures with the **Set up fixtures** function, only STL files can be used.

You can use the **3D mesh** function (option 152) to create STL files from other file types and adapt STL files to the requirements of your control.

Further information: "Generating STL files with 3D mesh (option 152)", Page 1372

Alternatively, CFG and M3D files can be set up manually.

Fixtures from STL files

STL files allow you to map both individual components and entire assemblies as an immobile fixture. The STL format is useful, in particular, for integrated fixture systems and recurring setups.

If an STL file does not meet the requirements of the control, then the control issues an error message.

With CAD Model Optimizer (software option 152), you can adapt STL files that do not meet the requirements and then use them as fixtures.

Further information: "Generating STL files with 3D mesh (option 152)", Page 1372

Fixtures from M3D files

M3D is a file type designed by HEIDENHAIN. The paid M3D Converter software from HEIDENHAIN allows you to create M3D files from STL or STEP files.

In order to use an M3D file as a fixture, you need to use the M3D Converter software to create and check the file.

Fixtures from CFG files

CFG files are configuration files. You can integrate the STL and M3D files available in a CFG file. This enables you to map complex setups.

The **Set up fixtures** function can be used to create a CFG file for the fixture, using the measured value.

In CFG files, you can correct the orientation of the fixture files to be effective on the control. **KinematicsDesign** can be used to create and edit CFG files on the control.

Further information: "Editing CFG files with KinematicsDesign", Page 1103

Notes

NOTICE

Danger of collision!

The setup situation defined for fixture monitoring must match the actual machine status. Otherwise, there is a risk of collision.

- Measure the position of the fixture in your machine
- Use the measured values for positioning the fixture
- Test the NC programs in the Simulation

- When using a CAM system, use a postprocessor to output the fixture situation.
- Note the orientation of the coordinate system in the CAD system. Use the CAD system to adapt the orientation of the coordinate system to the desired orientation of the fixture in the machine.
- You can choose any orientation of the fixture model in the CAD system, and therefore the orientation does not always match the orientation of the fixture in the machine.
- Define the coordinate origin in the CAD system such that the fixture can be directly attached to the point of insertion of the kinematics.
- Create a central directory for your fixtures (e.g., **TNC:\system\Fixture**).
- HEIDENHAIN recommends storing variants of recurring setup situations suitable for standard workpiece sizes in the control (e.g., vise with different jaw opening widths).

  By storing multiple fixtures, you can choose the appropriate fixture for your machining operation without needing to configure it.
- Example files for setups used in everyday manufacturing are provided in the NC database of the Klartext Portal:

  https://www.klartext-portal.com/en/tips/nc-solutions

18.2.2 Integrating the fixtures into collision monitoring (option 140)

Application

The **Simulation** function determines the position of a 3D model in the **Set up fixtures** workspace, matching the real fixture in the machine envelope. Once the fixture has been set-up, the control considers it in Dynamic Collision Monitoring (DCM).

Related topics

- **Simulation** workspace

  Further information: "Simulation Workspace", Page 1405
- Dynamic Collision Monitoring (DCM)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084
- Fixture monitoring

  Further information: "Fixture monitoring (option 40)", Page 1091

Requirements

- Dynamic Collision Monitoring (DCM version 2, software option 140)
- Workpiece touch probe
- Permitted fixture file matching the real fixture

  Further information: "Options for fixture files", Page 1092

Description of function

The **Set up fixtures** function is available as a touch probe function in the **Setup** application of the **Manual** operating mode.

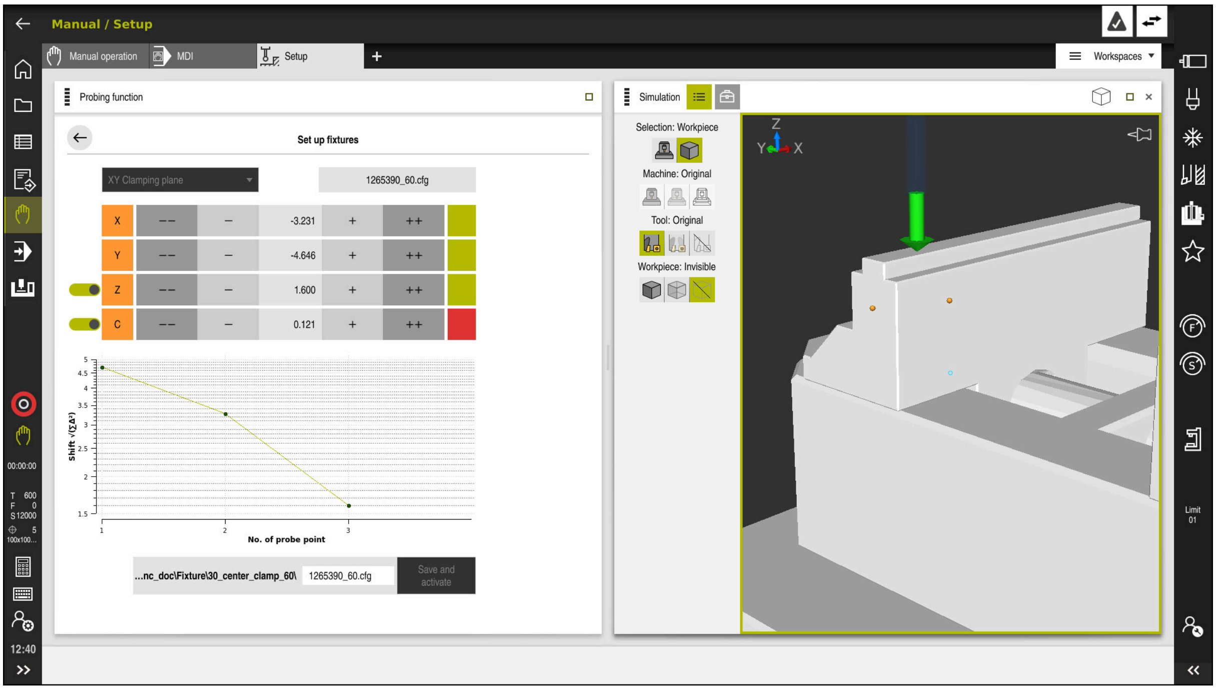
The **Set up fixtures** function determines the fixture position, using various probing processes. First one point on the fixture is probed in every linear axis. The position of the fixture is defined in this way. After probing one point in all linear axes, further points can be integrated in order to improve positioning accuracy. After defining the position in one axis, the control changes the status of that axis from red to green.

The revision diagram shows by which value the 3D model is shifted on the real fixture resulting from the individual probing processes.

Further information: "Revision diagram", Page 1098

Extensions of the Simulation workspace

In addition to the **Probing function** workspace, the **Simulation** workspace offers graphic support for setting up the fixture.



Set up fixtures function with open **Simulation** workspace

When the **Set up fixtures** function is active, the **Simulation** workspace shows the content below:

- Current position of fixture as viewed by the control
- Probed points on the fixture
- Possible direction of probing by means of an arrow:
  - No arrow

    Probing is not possible. The workpiece touch probe is too distant from the fixture or the workpiece touch probe is positioned within the fixture, as seen by the control.
  - Red arrow

    Probing in the direction of the arrow is not possible. The angle between the workpiece touch probe and the fixture is not correct.

    > Probing on edges, corners or heavily curved fixture areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.
  - Yellow arrow

    Probing in the direction of the arrow is possible. Probing will presumably not improve the set-up process, e. g. the probing point in the displayed axis direction is too close to an already probed point.
  - Green arrow

    Probing in the direction of the arrow is possible. The angle between the workpiece touch probe and the fixture is correct and the distance is right.

Icons and buttons

The **Set up fixtures** function contains the following icons and buttons:

Icon or button	Function
XY Clamping plane	This selection menu defines the plane in which the fixture is in contact with the machine. The control offers the following planes: ■ XY clamping plane ■ XZ clamping plane ■ YZ clamping plane Depending on the selected clamping plane, the control displays the axes in question. In the **XY Clamping plane**, the control displays e. g. the axes **X**, **Y**, **Z** and **C**.
127_Fixture.cfg	Name of fixture file
– –	Shifts the position of the virtual fixture by 10 mm or 10° in the negative axis direction Shifts the fixture in mm in a linear axis and in degrees in a rotary axis.
–	Shifts the position of the virtual fixture by 1 mm or 1° in the negative axis direction
0.000	Enter the position of the virtual fixture directly
+	Shifts the position of the virtual fixture by 1 mm or 1° in the positive axis direction
++	Shifts the position of the virtual fixture by 10 mm or 10° in the positive axis direction
	Status of axis The control displays the following colors: ■ Gray The axis is hidden in this set-up process and is not considered. ■ White At the beginning of the set-up process, i. e. before probing points are calculated, the control displays the status of all axes in white. ■ Red The position of the fixture in this axis is not unambiguously defined. ■ Yellow The position of the fixture in this axis already contains information. The information is not meaningful yet. ■ Green The position of the fixture in this axis is unambiguously defined.
...EMO_2021\Digital_Process_Chain\ 127_Fixture.cfg	Path of fixture file The control automatically saves the fixture file in the initial folder. The fixture file name can be edited.

Icon or button	Function
Save and activate	This function saves all obtained data in a CFG file and activates the measured fixture in Dynamic Collision Monitoring (DCM). When using a CFG file as the data source for the measuring process, the existing CFG file can be overwritten by **Save and activate** at the end of the measuring process. When creating a new CFG file, enter a different file name in the path.

When using a datum clamping system and for this reason, one axis (such as **Z**) is not to be considered when setting up the fixture, the axis in question can be deselected by a switch. The control will not take deselected axes into account in the set-up process and positions the fixture by considering the remaining axes only.

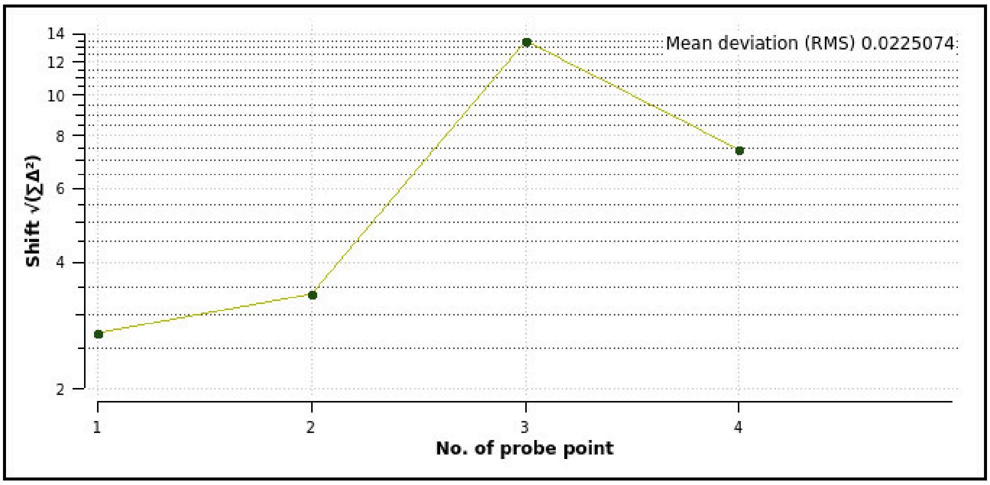
Revision diagram

Every performed probing restricts the possible positioning of the fixture more and more and puts the 3D model closer to the actual position in the machine.

The revision diagram shows the curve of revisions performed during the setting-up process. The set-up process is successfully completed when only revisions in the accuracy range of e. g. 0.05 mm appear.

The factors below influence the accuracy that can be achieved when measuring fixtures:

- Accuracy of workpiece touch probe
- Repeatability of workpiece touch probe
- Accuracy of 3D model
- Condition of the actual fixture, e. g. existing wear or score marks



Revision diagram in the **Set up fixtures** function

The revision diagram of the **Set up fixtures** function shows the following information:

- **Mean deviation (QMW)**

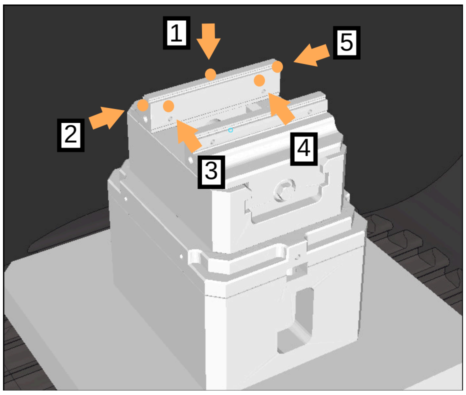
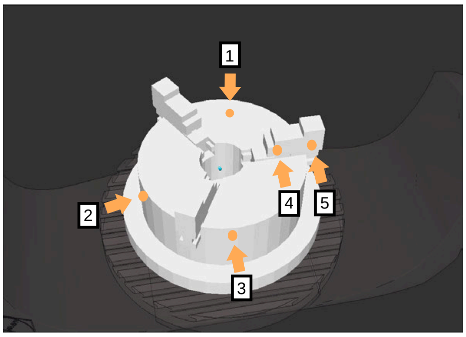
  This area shows the average distance of the measured probing points from the 3D model in mm.
- **Shift**

  This axis shows the course of the revised model position by means of the added probing points. The individual values show by which value the 3D model was shifted by the respective probing process.
- **No. of probe point**

  This axis shows the numbers of the individual probing points.

Example of sequence of fixture probing points

The following probing points can e. g. be set for different fixtures:

Chucking equipment/fixtures	Possible sequence
 Probing points for a vice with a fixed vice jaw	The following probing points can be set when measuring a vice: 1 Touching the fixed vice jaw in **Z-** 2 Touching the fixed vice jaw in **X+** 3 Touching the fixed vice jaw in **Y+** 4 Touching the second value in **Y+** for rotation 5 To improve accuracy, touching the check point in **X-**
 Probing points with a three-jaw chuck	The following probing points can be set when measuring a three-point chuck: 1 Touching the jaw chuck body in **Z-** 2 Touching the jaw chuck body in **X+** 3 Touching the jaw chuck body in **Y+** 4 Touching the jaw in **Y+** for rotation 5 Touching the second value at the jaw in **Y+** for rotation

Measuring the fixed-jaw vice

> The desired 3D model must meet the requirements of the control.
> **Further information:** "Options for fixture files", Page 1092

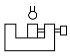
To measure a vice using the **Set up fixtures** function:

- Affix a real vice in the working space

- Select the **Manual** operating mode
- Insert the workpiece touch probe
- Manually position the workpiece touch probe above the fixed vice jaw at a striking point

> This step makes the subsequent steps easier.

- Select the **Setup** application



- Select **Set up fixtures**
- > The control opens the **Set up fixtures** menu.
- Select a 3D model matching the real vice

Open

- Select **Open**
- > The control opens the selected 3D model in the simulation.

++

- Pre-position the 3D model by using the buttons for the individual axes within the virtual working space

> For pre-positioning the vice, use the workpiece touch probe as a point of reference.
> At this point in time, the control does not know the precise position of the fixture, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe and by using e. g. the table's T-slots produces values close to the position of the real vice.
> Even after recording the first measuring points, the shifting functions are still available for correcting the fixture position manually.

- Specify the clamping plane, e. g. **XY**
- Position the workpiece touch probe until a green down arrow appears

> As the 3D model is only pre-positioned at this point in time, the green arrow cannot provide any reliable information about whether the desired surface of the fixture will actually be touched. Check if the fixture position in the simulation and in the machine match and if touching in the direction of the arrow is possible on the machine.
> Do not touch directly near edges, chamfers and roundings.

- Press the **NC Start** key
- > The control probes in the direction of the arrow.
- > The control displays the status of the **Z** axis in green and shifts the fixture to the touched position. The control marks the touched position by a point in the simulation.
- Repeat this process in axis directions **X+** and **Y+**
- > The status of the axes turns green.
- Touch another point in axis direction **Y+** for the basic rotation

> To achieve maximum accuracy when touching the basic rotation, the probing points should be as far apart from one another as possible.

- > The control changes the status of the **C** axis to green.
- Touch the check point in axis direction **X-**

> Additional check points at the end of the measuring process improve the matching accuracy and minimize the faults between the 3D model and the real fixture.

- Select **Save and activate**
- > The control closes the **Set up fixtures** function, saves a CFG file with the measured values at the path specified above, and integrates the measured fixture into Dynamic Collision Monitoring (DCM)

Notes

NOTICE

Danger of collision!

To touch the exact position of the fixture on the machine, the workpiece touch probe must be properly calibrated and the value **R2** properly defined in the tool management. Otherwise, incorrect tool data of the workpiece touch probe may cause inaccurate measurement and possibly a collision.

- Calibrate the workpiece touch probe at regular intervals
- Enter parameter **R2** in the tool management

- The control cannot identify modeling differences between the 3D model and the real fixture.
- At the time of set-up, Dynamic Collision Monitoring (DCM) does not know the exact position of the fixture. In this condition, collisions with the fixture, the tool or other non-machine components such as fixing clamps in the work envelope may occur. The non-machine components can be modeled on the control using a CFG file.

  Further information: "Editing CFG files with KinematicsDesign", Page 1103
- If you cancel the **Set up fixtures** function, DCM will not monitor the fixture. In this case, any fixtures previously set up are also removed from the scope of monitoring. The control displays a warning.
- Only one fixture can be measured at a time. To monitor several fixtures simultaneously by DCM, the fixtures must be integrated into a CFG file.

  Further information: "Editing CFG files with KinematicsDesign", Page 1103
- When measuring a jaw chuck, the coordinates of the axes **Z**, **X** and **Y** are determined just as when measuring a vice. The rotation is determined from one single jaw.
- The saved fixture file can be integrated into the NC program with the **FIXTURE SELECT** function. This can be used for simulating and executing the NC program, considering the real setup situation.

  Further information: "Loading and removing fixtures using the FIXTURE function (option 40)", Page 1102

18.2.3 Loading and removing fixtures using the FIXTURE function (option 40)

Application

The **FIXTURE** function allows loading and removing saved fixtures from within the NC program.

In the **Editor** operating mode and in the **MDI** application, different fixtures can be loaded independently of one another.

Further information: "Fixture monitoring (option 40)", Page 1091

Requirements

- Dynamic Collision Monitoring (DCM, software option 40)
- A measured fixture file exists

Description of function

The selected setup situation is checked for collisions during simulation or machining.

The **FIXTURE SELECT** function selects a fixture by means of a pop-up window. The search filter in the window may have to be changed to **All files (.*)**.

The **FIXTURE RESET** function removes the fixture.

Input

11 FIXTURE SELECT "TNC:\system \Fixture\JAW_CHUCK.STL"	; Load the fixture as an STL file

The NC function includes the following syntax elements:

Syntax element	Meaning
FIXTURE	Syntax initiator for fixtures
SELECT or **RESET**	Select or remove fixture
File or **QS**	Fixture path as a fixed or variable name Only if **SELECT** has been selected

18.2.4 Editing CFG files with KinematicsDesign

Application

KinematicsDesign allows editing CFG files in the control. In this process, **KinematicsDesign** displays the fixtures graphically and thus supports troubleshooting and removal of errors. Several fixtures can be joined in order to take complex clamping situations into account in Dynamic Collision Monitoring (DCM).

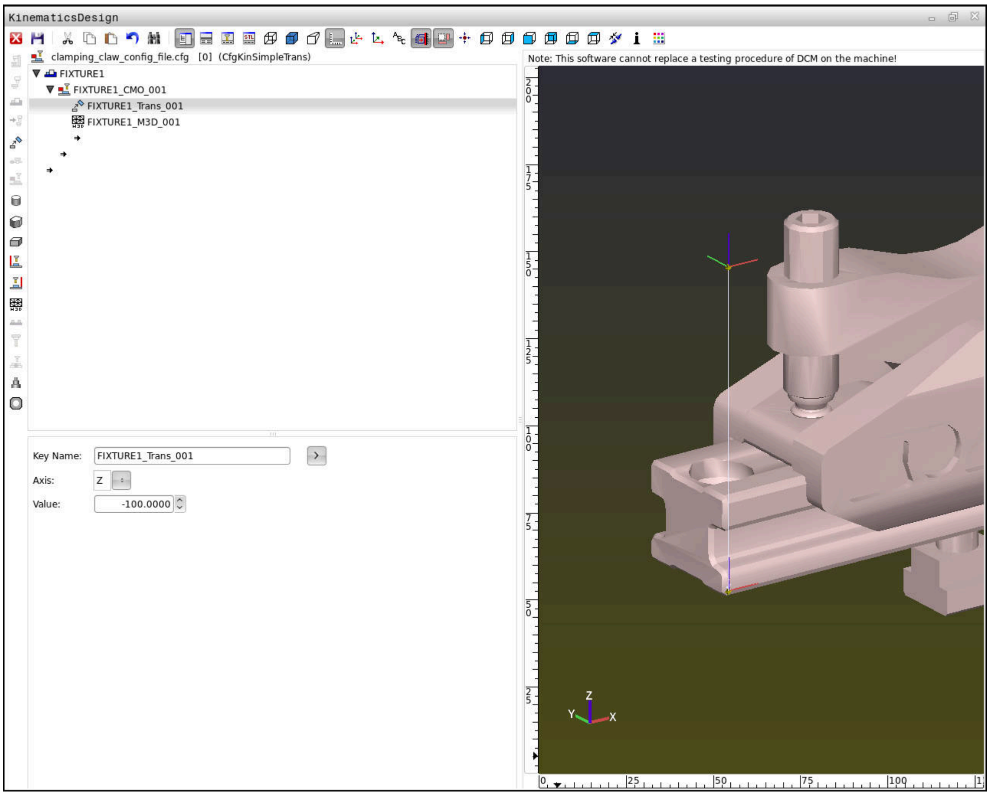
Description of function

When preparing a CFG file in the control, the control automatically opens the file with **KinematicsDesign**.

KinematicsDesign offers the following functions:

- Editing of fixtures with graphic support
- Feedback in case of incorrect entries
- Integration of transformations
- Addition of new elements
  - 3D model (M3D or STL files)
  - Cylinder
  - Prism
  - Cuboid
  - Truncated cone
  - Hole

You can integrate both STL files and M3D files into CFG files more than once.



Syntax in CFG files

The following syntax elements are used within the various CFG functions:

Function	Description
`key:= ""`	Name of the function
`dir:= ""`	Direction of a transformation (e.g., **X**)
`val:= ""`	Value
`name:= ""`	Name displayed if a collision occurs (optional input)
`filename:= ""`	File name
`vertex:= [ ]`	Position of a cube
`edgeLengths:= [ ]`	Dimensions of a cuboid
`bottomCenter:= [ ]`	Center of a cylinder
`radius:=[ ]`	Radius of a cylinder
`height:= [ ]`	Height of a geometric object
`polygonX:= [ ]`	Line of a polygon in X
`polygonY:= [ ]`	Line of a polygon in Y
`origin:= [ ]`	Starting point of a polygon

Each element is assigned its own **key**. A **key** must be unambiguous and unique, meaning that it must not occur more than once in the description of a fixture. Based on the **key**, the elements are referenced to each other.

The following functions are available if you wish to use CFG functions to describe a fixture in the control:

Function	Description
`CfgCMOMesh3D(key:="Fixture_body", filename:="1.STL",name:="")`	Definition of fixture component You can also enter an absolute path for the defined fixture component (e.g., **TNC:\nc_prog\1.STL**)
`CfgKinSimpleTrans(key:="XShiftFixture", dir:=X,val:=0)`	Shift in X axis Inserted transformations, such as a shift or rotation, are effective for all of the elements following in the kinematic chain.
`CfgKinSimpleTrans(key:="CRot0", dir:=C,val:=0)`	Rotation in C axis
`CfgCMO ( key:="fixture", primitives:= ["XShiftFixture","CRot0", "Fixture_body"], active :=TRUE, name :="")`	Describes all of the transformations contained in the fixture. The parameter active **:= TRUE** activates collision monitoring for the fixture. The **CfgCMO** contains collision objects and transformations. The fixture is combined based on the arrangement of the different transformations. Here, the transformation **XShiftFixture** shifts the center of rotation of the transformation **CRot0**.

Function	Description
`CfgKinFixModel(key:="Fix_Model", kinObjects:=["fixture"])`	Fixture designation **CfgKinFixModel** contains one or more **CfgCMO** elements.

Geometric shapes

You can add simple geometric objects to your collision object either directly in the CFG file or by using **KinematicsDesign**.

All integrated geometric shapes are subelements of the higher-order **CfgCMO**, in which they are listed as **primitives**.

The following geometric objects are available:

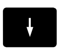
Function	Description
`CfgCMOCuboid ( key:="FIXTURE_Cub", vertex:= [ 0, 0, 0 ], edgeLengths:= [0, 0, 0], name:="" )`	Definition of a cuboid
`CfgCMOCylinder ( key:="FIXTURE_Cyl", dir:=Z, bottomCenter:= [0, 0, 0], radius:=0, height:=0, name:="")`	Definition of a cylinder
`CfgCMOPrism ( key:="FIXTURE_Pris_002", height:=0,  polygonX:=[],  polygonY:=[], name:="",  origin:= [ 0,  0,  0 ] )`	Definition of a prism A prism is described by entering the height and several polygonal lines.

Creating a fixture entry with a collision object

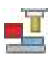
The content below describes the procedure with **KinematicsDesign** opened.

To create a fixture entry with a collision object:

- ▶ Select **Insert fixture**
- > **KinematicsDesign** creates a new fixture entry within the CFG file.
- ▶ Enter a **keyname** for the fixture (e.g., **clamping jaw**)
- ▶ Confirm your input
- > **KinematicsDesign** loads the input.



- ▶ Move cursor down one level



- ▶ Select **Insert collision object**
- ▶ Confirm your input
- > **KinematicsDesign** creates a new collision object.

Defining geometric shapes

KinematicsDesign allows you to define various geometric shapes. You can construct simple fixtures by combining several geometric shapes.

To define a geometric shape:

- Create a fixture entry with a collision object

⇒
- Select the cursor key beneath the collision object

- Select the desired geometric shape (e.g., a cuboid)
- Define the position of the cuboid (e.g., **X = 0**, **Y = 0**, **Z = 0**)
- Define the dimensions of the cuboid (e.g., **X = 100**, **Y = 100**, **Z = 100**)
- Confirm your input
- > The control displays the defined cuboid in the graphic.

Integrating 3D models

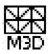
The integrated 3D models must meet the requirements of the control.

Further information: "Options for fixture files", Page 1092

To integrate a 3D model as a fixture:

- Create a fixture entry with a collision object

⇒
- Select the cursor key beneath the collision object



- Select **Insert 3D model**
- > The control opens the **Open file** window.
- Select the desired STL or M3D file
- Press **OK**
- > The control integrates the selected file and displays the file in the graphic window.

Fixture placement

You can place the integrated fixture at any position (e.g., for correcting the orientation of an external 3D model). For this purpose, insert transformations for all axes you wish to use.

To position a fixture with **KinematicsDesign**:

- Define the fixture

⇒
- Select the cursor key beneath the element to be positioned

- Select **Insert transformation**
- Enter a **key name** for the transformation (e.g., **Z shift**)
- Select the **axis** for the transformation (e.g., **Z**)
- Select the **value** for the transformation (e.g., **100**)
- Confirm your input
- > **KinematicsDesign** inserts the transformation.
- > **KinematicsDesign** depicts the transformation in the graphic.

Note

As an alternative to using **KinematicsDesign**, you can also create fixture files directly from the CAM system or by using the appropriate code in a text editor.

Example

The example below describes the syntax of a CFG file for a vise with two movable jaws.

Files used

Various STL files are used to describe the vise. Since the jaws of the vise are dimensionally identical, they are defined using the same STL file.

Code	Explanation
`CfgCMOMesh3D (key:="Fixture_body", filename:="vice_47155.STL", name:="")`	Body of the vise
`CfgCMOMesh3D (key:="vice_jaw_1", filename:="vice_jaw_47155.STL", name:="")`	First jaw of the vise
`CfgCMOMesh3D (key:="vice_jaw_2", filename:="vice_jaw_47155.STL", name:="")`	Second jaw of the vise

Definition of jaw opening width

In this example, the opening width of the vise is defined using two mutually dependent transformations.

Code	Explanation
`CfgKinSimpleTrans (key:="TRANS_opening_width", dir:=Y, val:=-60)`	Jaw opening width of the vise in Y direction: 60 mm
`CfgKinSimpleTrans (key:="TRANS_opening_width_2", dir:=Y, val:=30)`	Position of the fist jaw of the vise in Y direction: 30 mm

Positioning of the fixture within the working space

The defined fixture components are positioned using various transformations.

Code	Explanation

`CfgKinSimpleTrans (key:="TRANS_X", dir:=X, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Y", dir:=Y, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Z", dir:=Z, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_Z_vice_jaw", dir:=Z, val:=60)`
`CfgKinSimpleTrans (key:="TRANS_C_180", dir:=C, val:=180)`
`CfgKinSimpleTrans (key:="TRANS_SPC", dir:=C, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_SPB", dir:=B, val:=0)`
`CfgKinSimpleTrans (key:="TRANS_SPA", dir:=A, val:=0)`
	Positioning of the fixture components
	In this example, a rotation by 180° is inserted for rotating the defined jaw of the vise. This is necessary because the same initial model is used for both jaws of the vise.
	The rotation inserted applies to all components following in the transformation chain.

Description of the fixture

You need to combine all objects and transformations in the CFG file in order to ensure that the fixture is correctly depicted in the simulation.

Code	Explanation
`CfgCMO (key:="FIXTURE", primitives:= [ "TRANS_X", "TRANS_Y", "TRANS_Z", "TRANS_SPC", "TRANS_SPB", "TRANS_SPA", "Fixture_body", "TRANS_Z_vice_jaw", "TRANS_opening_width_2", "vice_jaw_1", "TRANS_opening_width", "TRANS_C_180", "vice_jaw_2" ], active:=TRUE, name:="")`	Combining the transformations and objects contained in the fixture

Fixture designation

You need to assign a designation to the combined fixture.

Code	Explanation
`CfgKinFixModel (key:="FIXTURE1", kinObjects:=["FIXTURE"])`	Designation of the combined fixture

18.3 Advanced checks in the simulation

Application

The **Advanced checks** function allows checking in the **Simulation** workspace if collisions occur between the workpiece and the tool or tool carrier.

Related topics

- Collision monitoring of machine component by Dynamic Collision Monitoring (DCM, option 40)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084

Description of function

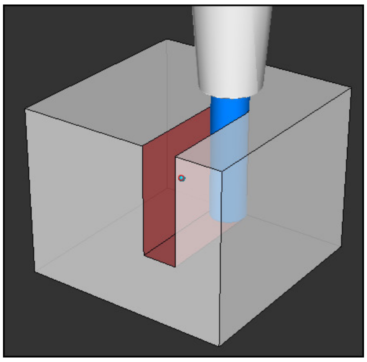
The **Advanced checks** function can be used only in the **Editor** operating mode.

The **Advanced checks** function can be activated by a switch in the **Visualization options** column.

Further information: "Visualization options column", Page 1407

If the **Advanced checks** function is active, the control generates a warning in the cases below:

- Material removal at rapid traverse
  The control displays material removal at rapid traverse in red in the simulation.
- Collisions between the tool and the workpiece
- Collisions between the tool holder and the workpiece
  The control also considers inactive steps of a stepped tool.



Material removal at rapid traverse

Notes

- The **Advanced checks** function helps reduce the danger of collision. However, the control cannot consider all possible constellations during operation.
- The **Advanced checks** function in the simulation uses the information from the workpiece blank definition for workpiece monitoring. The control can monitor only the active workpiece blank, even if several workpieces are clamped in the machine!
  Further information: "Defining a workpiece blank with BLK FORM", Page 234

18.4 Automatic tool liftoff with FUNCTION LIFTOFF

Application

The tool retracts from the contour by up to 2 mm. The control calculates the lift off direction based on the input in the **FUNCTION LIFTOFF** block.

The **LIFTOFF** function is effective in the following situations:

- In case of an NC stop triggered by you
- In case of an NC stop triggered by the software, e. g. if an error has occurred in the drive system.
- In case of a power interruption

Related topics

- Automatic liftoff with **M148**

  Further information: "Automatically lifting off upon an NC stop or a power failure with M148", Page 1257
- Liftoff in the tool axis with **M140**

  Further information: "Retracting in the tool axis with M140", Page 1252

Requirements

- Function enabled by the machine manufacturer

  In machine parameter **on** (no. 201401), the machine manufacturer defines whether automatic lift-off is active.
- **LIFTOFF** activated for the tool

  You must define the value **Y** in the **LIFTOFF** column of the tool management.

Description of function

You can program the LIFTOFF function in the following ways:

- **FUNCTION LIFTOFF TCS X Y Z**: Lift-off in the tool coordinate system (**T-CS**) with the vector resulting from **X**, **Y** and **Z**
- **FUNCTION LIFTOFF ANGLE TCS SPB**: Lift-off in the tool coordinate system (**T-CS**) with a defined spatial angle

  This makes sense for turning (option 50)
- **FUNCTION LIFTOFF AUTO**: Liftoff in automatically determined direction
- **FUNCTION LIFTOFF RESET**: NC function reset

Further information: "Tool coordinate system T-CS", Page 946

The control automatically resets the **FUNCTION LIFTOFF** function at the end of a program.

FUNCTION LIFTOFF in turning mode (option 50)

NOTICE

Caution: Danger to the tool and workpiece!

Undesired movements of the axes can occur if you use the **FUNCTION LIFTOFF ANGLE TCS** function in turning mode. The behavior of the control depends on the kinematics description and Cycle **800** (**Q498 = 1**).

- ▶ Carefully test the NC program or program section in **Program run, single block** operating mode.
- ▶ If necessary, change the algebraic sign of the defined angle

If parameter **Q498** has been set to 1, the control will reverse the tool for machining.

In conjunction with the **LIFTOFF** function, the control behaves as follows:

- If the tool spindle has been defined as an axis, the **LIFTOFF** direction will be reversed.
- If the tool spindle has been defined as a kinematic transformation, the **LIFTOFF** direction will not be reversed.

Further information: "Cycle 800 ADJUST XZ SYSTEM ", Page 681

Input

11 FUNCTION LIFTOFF TCS X+0 Y+0.5 Z +0.5	; Lift off with the defined vector upon NC stop or power failure
12 FUNCTION LIFTOFF ANGLE TCS SPB +20	; Lift off with spatial angle **SPB** +20 upon NC stop or power failure

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION LIFTOFF	Syntax initiator for a automatic liftoff
TCS, **ANGLE**, **AUTO** or **RESET**	Define the liftoff direction as a vector, define it as a spatial angle, have it calculated automatically, or reset liftoff
X, **Y**, **Z**	Vector components in the tool coordinate system **T-CS** Only if **TCS** has been selected
SPB	Spatial angle in **T-CS** Only if **ANGLE** has been selected When entering 0, the control lifts off in the direction of the active tool axis.

Notes

- The control uses the **M149** function to deactivate the **FUNCTION LIFTOFF** function without resetting the lift-off direction. If you program **M148**, the control will activate the automatic lift-off of the tool in the lift-off direction defined by the **FUNCTION LIFTOFF** function.
- In case of an emergency stop, the control will not lift off the tool.
- The lift-off movement will not be monitored by Dynamic Collision Monitoring (DCM, option 40)

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084
- In machine parameter **distance** (no. 201402), the machine manufacturer defines the maximum lift-off height.

19

Control Functions

19.1 Adaptive Feed Control (AFC, option 45)

19.1.1 Fundamentals

Application

Adaptive Feed Control (AFC) saves time when processing NC programs and reduces wear on the machine. The control regulates the contouring feed rate during program run depending on the spindle power. In addition, the control responds to overloading of the spindle.

Related topics

- Tables related to AFC

  Further information: "Tables for AFC (option 45)", Page 1872

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Approved by the machine manufacturer

  The machine manufacturer uses the optional machine parameter **Enable** (no. 120001) to define whether you can use AFC.

Description of function

To regulate the feed rate during program run with AFC:

- Define basic settings for AFC in the **AFC.tab** table

  Further information: "Basic AFC settings in AFC.tab", Page 1872
- Define settings for AFC for each tool in the tool management

  Further information: "Tool table tool.t", Page 1814
- Define AFC in the NC program

  Further information: "NC functions for AFC (option 45)", Page 1116
- Define AFC in the **Program Run** operating mode with the **AFC** switch.

  Further information: "AFC switch in Program Run operating mode", Page 1118
- Prior to automatic control, determine the reference spindle power with a teach-in cut

  Further information: "AFC teach-in cut", Page 1119

If AFC is active in the teach-in cut or in control mode, the control displays an icon in the **Positions** workspace.

Further information: "Positions workspace", Page 141

Detailed information about the function is provided by the control on the **AFC** tab of the **Status** workspace.

Further information: "AFC tab (option 45)", Page 150

Benefits of AFC

Adaptive feed control (AFC) has the following advantages:

- Optimization of machining time

  By controlling the feed rate, the control tries to maintain the previously recorded maximum spindle power or the reference power specified in the tool table (**AFC-LOAD** column) during the entire machining time. It shortens the machining time by increasing the feed rate in machining zones with little material removal.
- Tool monitoring

  If the spindle power exceeds the taught-in or specified maximum value, the control reduces the feed until the reference spindle power is reached. If the minimum feed rate is exceeded, the control executes a shutdown response. AFC can also use the spindle power to monitor the tool for wear and breakage without changing the feed rate.

  Further information: "Monitoring tool wear and tool load", Page 1120
- Protection of the machine’s mechanical elements

  Timely feed rate reduction and shutdown responses help to avoid machine overload.

Tables related to AFC

The control offers the following tables in conjunction with AFC:

- **AFC.tab**

  In the **AFC.TAB** table, you can enter the feed rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.

  Further information: "Basic AFC settings in AFC.tab", Page 1872
- ***.H.AFC.DEP**

  With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. **<name>** is the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value to the table.

  Further information: "AFC.DEP settings file for teach-in cuts", Page 1875
- ***.H.AFC2.DEP**

  During a teach-in cut, the control stores information for each machining step in the **<name>.H.AFC2.DEP** file. The **<name>** corresponds to the name of the NC program for which you perform the teach-in cut.

  In control mode, the control updates the data in this table and performs evaluations.

  Further information: "Log file AFC2.DEP", Page 1876

You can open and, if necessary, edit the tables for AFC during program run. The control offers only the tables for the active NC program.

Further information: "Editing tables for AFC", Page 1878

Notes

NOTICE

Caution: Danger to the tool and workpiece!

As soon as Adaptive Feed Control (AFC) is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate, e.g. due to wear, before it was deactivated, the control accelerates the feed rate up to the programmed value. This behavior applies regardless of how the function is deactivated. This feed acceleration may result in damage to the tool and/or the workpiece!

- If the feed rate falling below the **FMIN** value is imminent, stop the machining operation without deactivating AFC
- Define the overload reaction for cases in which the feed rate falls below the **FMIN** value

- If Adaptive Feed Control is active in **Control** mode, the control executes a shutdown response independent of the programmed overload reaction.
  - If, with the reference spindle load, the minimum feed factor is fallen below

    The control executes the shutdown response from the **OVLD** column of the **AFC.tab** table.

    Further information: "Basic AFC settings in AFC.tab", Page 1872
  - If the programmed feed rate falls below the 30 % threshold

    The control executes an NC stop.
- Adaptive feed control is not intended for tools with diameters less than 5 mm. If the rated power consumption of the spindle is very high, the limit diameter of the tool may be larger.
- Do not work with adaptive feed control in operations in which the feed rate and spindle speed must be adapted to each other, such as tapping.
- In NC blocks containing **FMAX**, the adaptive feed control is **not active**.
- With the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control displays dependency files in the file management.

19.1.2 Activating and deactivating AFC

NC functions for AFC (option 45)

Application

Adaptive Feed Control (AFC) is activated and deactivated from the NC program.

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Control settings defined in the **AFC.tab** table

  Further information: "Basic AFC settings in AFC.tab", Page 1872
- Desired control setting defined for all tools

  Further information: "Tool table tool.t", Page 1814
- **AFC** switch active

  Further information: "AFC switch in Program Run operating mode", Page 1118

Description of function

The control provides several functions that enable you to start and stop AFC:

- **FUNCTION AFC CTRL**: The **AFC CTRL** function activates feedback control mode starting with this NC block, even if the learning phase has not been completed yet.
- **FUNCTION AFC CUT BEGIN TIME1 DIST2 LOAD3**: The control starts a sequence of cuts with active **AFC**. The changeover from the teach-in cut to feedback control mode begins as soon as the reference power has been determined in the teach-in phase, or once one of the **TIME**, **DIST** or **LOAD** conditions has been met.
- **FUNCTION AFC CUT END**: The **AFC CUT END** function deactivates the AFC control.

Input

FUNCTION AFC CTRL

11 FUNCTION AFC CTRL	; Start AFC in control mode

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION AFC CTRL	Syntax initiator for the start of control mode

FUNCTION AFC CUT

11 FUNCTION AFC CUT BEGIN TIME10 DIST20 LOAD80	; Start AFC machining step, limit the duration of the teach-in phase

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION AFC CUT	Syntax initiator for an AFC machining step
BEGIN or **END**	Start or end machining step
TIME	End teach-in phase after the defined time in seconds Optional syntax element Only if **BEGIN** has been selected
DIST	End teach-in phase after the defined distance in mm Optional syntax element Only if **BEGIN** has been selected
LOAD	Enter the reference load of the spindle directly, max. 100% Optional syntax element Only if **BEGIN** has been selected

Notes

NOTICE
Caution: Danger to the tool and workpiece!
If you activate the **FUNCTION MODE TURN** machining mode, the control will clear the current **OVLD** values. This means that you need to program the machining mode before the tool call! If the programming sequence is not correct, no tool monitoring will take place, which might result in damage to the tool or workpiece!
▶ Program the **FUNCTION MODE TURN** machining mode before the tool call

- The **TIME**, **DIST** and **LOAD** defaults are modally effective. They can be reset by entering **0**.
- Execute the function **AFC CUT BEGIN** only after the starting rotational speed has been reached. If this is not the case, then the control issues an error message, and the AFC cut is not started.
- You can define a feedback-control reference power with the **AFC LOAD** tool table column and the **LOAD** input in the NC program. You can activate the **AFC LOAD** value via the tool call and the **LOAD** value with the **FUNCTION AFC CUT BEGIN** function.

  If you program both values, the control will use the value programmed in the NC program!

AFC switch in Program Run operating mode

Application

The **AFC** switch allows you to activate or deactivate Adaptive Feed Control (AFC) in the **Program Run** operating mode.

Related topics

- Activating AFC in the NC program

  Further information: "NC functions for AFC (option 45)", Page 1116

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Approved by the machine manufacturer

  The machine manufacturer uses the optional machine parameter **Enable** (no. 120001) to define whether you can use AFC.

Description of function

The **AFC** switch must be activated for the NC functions to have an effect for AFC.

If you do not specifically deactivate AFC using the switch, AFC remains active. The control remembers the setting of the switch even if the control is restarted.

If the **AFC** switch is active, the control displays an icon in the **Positions** workspace. In addition to the current setting of the feed rate potentiometer, the control shows the controlled feed value as a percentage (%).

Further information: "Positions workspace", Page 141

Notes

NOTICE

Caution: Danger to the tool and workpiece!

As soon as the AFC function is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate (e.g. due to wear) before it was deactivated, the control accelerates the feed rate up to the programmed value. This applies regardless of how the function is deactivated (e.g. feed rate potentiometer). This acceleration may result in damages to the tool or the workpiece!

- If it is imminent that the feed rate falls below the **FMIN** value, stop the machining operation (instead of deactivating the **AFC** function)
- Define the overload reaction for cases in which the feed rate falls below the **FMIN** value

- If Adaptive Feed Control is active in **Control** mode, the control internally sets the spindle override to 100 %. This means that you can no longer change the spindle speed.
- If Adaptive Feed Control is active in **Control** mode, the control takes over the value from the feed rate override function.
  - Increasing the feed rate override has no influence on the control.
  - If you reduce the feed override with the potentiometer by more than 10% in relation to the position at the start of the program, the control switches AFC off.

    You can reactivate control with the **AFC** switch.
  - Potentiometer values of up to 50% always have an effect, even with active control.
- Mid-program startup is allowed during active feed control. The control takes the cutting number of the startup block in account.

19.1.3 AFC teach-in cut

Application

With the teach-in cut, the control determines the reference power of the spindle for the machining step. Based on the reference power, the control adjusts the feed rate in control mode.

If you have already determined the reference power for a machining operation, you can specify the value for the machining operation. For this, the control provides the **AFC-LOAD** column in the tool management and the **LOAD** syntax element in the **FUNCTION AFC CUT BEGIN** function. In this case, the control no longer performs a teach-in cut, but uses the specified value immediately for control.

Related topics

- Enter the known reference power in the **AFC-LOAD** column in the tool management

  Further information: "Tool table tool.t", Page 1814
- Define the known reference power in the **FUNCTION AFC CUT BEGIN** function

  Further information: "NC functions for AFC (option 45)", Page 1116

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Control settings defined in the **AFC.tab** table
  Further information: "Basic AFC settings in AFC.tab", Page 1872
- Desired control setting defined for all tools
  Further information: "Tool table tool.t", Page 1814
- Desired NC program selected in the **Program Run** operating mode
- **AFC** switch active
  Further information: "AFC switch in Program Run operating mode", Page 1118

Description of function

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**.

Further information: "AFC.DEP settings file for teach-in cuts", Page 1875

When you are performing a teach-in cut, the control shows the spindle reference power determined until this time in a pop-up window.

When the control has determined the control reference power, it ends the teach-in cut and switches to control mode.

Notes

- When you record a teach-in cut, the control internally sets the spindle override to 100%. Then you can no longer change the spindle speed.
- During the teach-in cut, you can influence the measured reference load by using the feed rate override to make any changes to the contouring feed rate.
- You can repeat a teach-in cut as often as desired. Manually change the status from **ST** back to **L**. If the programmed feed rate value is far too high and forces you to sharply decrease the feed rate override during the machining step, you will have to repeat the teach-in cut.
- If the determined reference load is greater than 2 %, the control changes the status from teach-in (**L**) to controlling (**C**). Adaptive feed control is not possible for smaller values.
- In **FUNCTION MODE TURN** machining mode, the minimum reference load is 5 %. Even if the control determines lower values, it will still use this minimum reference load. Thus, the overload limits (indicated as percentage values) are based on a minimum reference load of at least 5 %.

19.1.4 Monitoring tool wear and tool load

Application

With Adaptive Feed Control (AFC), you can monitor the tool for wear and breakage. The **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management can be used for this.

Related topics

- **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management
  Further information: "Tool table tool.t", Page 1814

Description of function

If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active.

Further information: "Basic AFC settings in AFC.tab", Page 1872

Tool wear monitoring

Activate cut-related tool wear monitoring by entering a value not equal to 0 in the **AFC-OVLD1** column in the tool table.

The shutdown response depends on the **AFC.TAB** column **OVLD**.

In conjunction with cut-related tool wear monitoring, the control only evaluates the options **M**, **E**, and **L** in the **OVLD** column. The following responses are possible:

- Pop-up window
- Lock current tool
- Insert replacement tool

Tool load monitoring

Activate cut-related tool load monitoring (tool breakage control) by entering a value not equal to 0 in the **AFC-OVLD2** column in the tool table.

As shutdown response, the control always executes a machining stop and locks the momentary tool.

In turning mode, the control can check for tool wear and tool breakage.

A tool breakage leads to a sudden load decrease. If you want the control to monitor the load decrease, too, enter the value 1 in the SENS column.

Further information: "Basic AFC settings in AFC.tab", Page 1872

19.2 Active Chatter Control (ACC, option 145)

Application

Chatter marks can be caused during heavy-duty machining, in particular. **ACC** reduces chattering, thereby reducing wear on the tool and machine. In addition, **ACC** increases metal removal rates.

Related topics

- ACC column in the tool table

  Further information: "Tool table tool.t", Page 1814

Requirements

- Active Chatter Control (ACC, software option 145)
- Control adapted by the machine manufacturer
- **ACC** column in the tool management defined with **Y**
- Number of tool cutting edges defined in the **CUT** column

Description of function

Strong forces come into play during roughing (power milling). Depending on the tool spindle speed, the resonances in the machine tool and the chip volume (metal-removal rate during milling), the machine can sometimes begin to **chatter**. This chattering places heavy strain on the machine, and causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can result in tool breakage.

In order to reduce a machine's tendency to chatter, HEIDENHAIN offers an effective control function known as Active Chatter Control (**ACC**). The use of this control function is particularly advantageous during heavy machining. ACC makes substantially higher metal removal rates possible. Depending on the type of machine, the metal-removal rate can often be increased by more than 25 %. You reduce the mechanical load on the machine and increase the life of your tools at the same time.

ACC was developed especially for roughing and heavy machining and is particularly effective in this area. You need to conduct appropriate tests to see whether ACC will also be advantageous on your machine and with your tool.

ACC is activated and deactivated using the **ACC** switch in the **Program Run** operating mode or the **MDI** application.

Further information: "Program Run operating mode", Page 1778

Further information: "Application MDI", Page 1759

If ACC is active, the control shows a corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 141

Notes

- ACC reduces or prevents vibrations in the range of 20 Hz to 150 Hz. If ACC does not appear to have an effect, the vibrations may be outside of this range.
- With Machine Vibration Control (MVC, software option 146), you can also positively influence the result.

19.3 Functions for controlling program run

19.3.1 Overview

The control provides the following NC functions for program control:

Syntax	Function	Further information
FUNCTION S-PULSE	Program pulsing spindle speed	Page 1122
FUNCTION DWELL	Program singular dwell time	Page 1123
FUNCTION FEED DWELL	Program cyclic dwell time	Page 1124

19.3.2 Pulsing spindle speed with FUNCTION S-PULSE

Application

Using the **S-PULSE FUNCTION** you can program a pulsing spindle speed, e.g. to avoid natural oscillations of the machine when operating at a constant spindle speed.

Description of function

With the **P-TIME** input value you define the duration of an oscillation (oscillation period), and with the **SCALE** input value the spindle speed change in percent. The spindle speed changes in a sinusoidal form around the target value.

Use **FROM-SPEED** and **TO-SPEED** to define the upper and lower spindle speed limits of a spindle speed range in which the pulsing spindle speed is effective. Both input values are optional. If you do not define a parameter, the function applies to the entire speed range.

Use the **FUNCTION S-PULSE RESET** to reset the pulsing spindle speed.

When a pulsing spindle speed is active, the control shows a corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 141

Input

11 FUNCTION S-PULSE P-TIME10 SCALE5 FROM-SPEED4800 TO-SPEED5200	; Spindle speed variation of 5 % around the nominal value within 10 seconds (with limit values)

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION S-PULSE	Start of syntax for pulsing spindle speed
P-TIME or **RESET**	Define the duration of an oscillation in seconds, or reset the pulsing spindle speed
SCALE	Spindle speed change in % Only if **P-TIME** has been selected
FROM-SPEED	Lower speed limit from which the pulsing spindle speed will be effective Only if **P-TIME** has been selected Optional syntax element
TO-SPEED	Upper speed limit up to which the pulsing spindle speed will be effective Only if **P-TIME** has been selected Optional syntax element

Note

The control never exceeds a programmed speed limit. The spindle speed is maintained until the sinusoidal curve of the **S-PULSE FUNCTION** falls below the maximum speed once more.

19.3.3 Programmed dwell time with FUNCTION DWELL

Application

The **FUNCTION DWELL** function enables you to program a dwell time in seconds or define the number of spindle revolutions for dwelling.

Related topics

- Cycle **9 DWELL TIME**

  Further information: "Cycle 9 DWELL TIME ", Page 1126
- Program recurring dwell time

  Further information: "Cyclic dwell time with FUNCTION FEED DWELL", Page 1124

Description of function

The defined dwell time from **FUNCTION DWELL** is effective in both milling and turning operations.

Input

11 FUNCTION DWELL TIME10	; Dwell time for 10 seconds
12 FUNCTION DWELL REV5.8	; Dwell time for 5.8 spindle revolutions

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION DWELL	Syntax initiator for singular dwell time
TIME or **REV**	Duration of dwell time in seconds or spindle revolutions

19.3.4 Cyclic dwell time with FUNCTION FEED DWELL

Application

The **FUNCTION FEED DWELL** function can be used to program a cyclic dwell time in seconds, e.g. to force chip breaking in a turning cycle.

Related topics

- Program a one-time dwell time

  Further information: "Programmed dwell time with FUNCTION DWELL", Page 1123

Description of function

The defined dwell time from **FUNCTION FEED DWELL** is effective in both milling and turning operations.

The **FUNCTION FEED DWELL** function is not effective with rapid traverse movements and probing motion.

Use **FUNCTION FEED DWELL RESET** to reset the recurring dwell time.

The control automatically resets the **FUNCTION FEED DWELL** function at the end of a program.

Program **FUNCTION FEED DWELL** immediately prior to the operation you wish to run with chip breaking. Reset the dwell time immediately following the machining with chip breaking.

Input

11 FUNCTION FEED DWELL D-TIME0.5 F-TIME5	; Activate cyclic dwell time: Machine for 5 seconds, hold for 0.5 seconds

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FEED DWELL	Syntax initiator for cyclic dwell time
D-TIME or **RESET**	Define dwell time duration in seconds or reset recurring dwell time
F-TIME	Duration of machining time until the next dwell time in seconds Only if **D-TIME** is selected

Notes

NOTICE

Caution: Danger to the tool and workpiece!

When the **FUNCTION FEED DWELL** function is active, the control will repeatedly interrupt the feed movement. While the feed movement is interrupted, the tool remains at its current position, and the spindle continues to turn. During thread cutting, this behavior will cause the workpiece to become scrap. There is also a risk of tool breakage during execution!

- Deactivate the **FUNCTION FEED DWELL** function before cutting threads

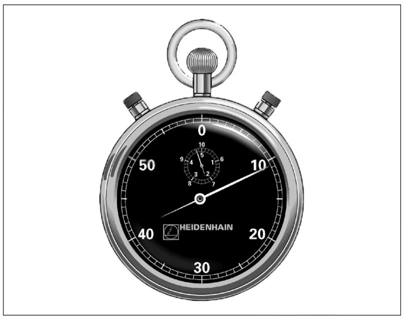
- You can also reset the dwell time by entering **D-TIME 0**.

19.4 Cycles with control function

19.4.1 Cycle 9 DWELL TIME

Application

This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.



Execution of the program run is delayed by the programmed **DWELL TIME**. A dwell time can be used for purposes such as chip breaking.

The cycle becomes effective as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.

Example

```
89 CYCL DEF 9.0  DWELL TIME
90 CYCL DEF 9.1  DWELL 1.5
```

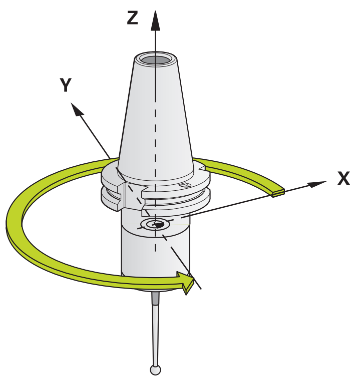
Cycle parameters

Help graphic	Parameter
	Dwell time in secs.? Enter the dwell time in seconds. Input: **0...3600 s** (**1** hour) in steps of 0.001 seconds

19.4.2 Cycle 13 ORIENTATION

Application

⚙	Refer to your machine manual. Machine and control must be specially prepared by the machine manufacturer for use of this cycle.



The control can control the main machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for purposes such as:

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

With **M19** or **M20**, the control positions the spindle at the angle of orientation defined in the cycle (depending on the machine).

If you program **M19** or **M20** without having defined Cycle **13** beforehand, the control positions the main spindle at an angle that has been set by the machine tool builder.

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- Cycle **13** is used internally for Cycles **202**, **204**, and **209**. Please note that, if required, you must program Cycle **13** again in your NC program after one of the machining cycles mentioned above.

Cycle parameters

Help graphic	Parameter
	Orientation angle Enter the angle relative to the angle reference axis of the working plane. Input: **0...360**

Example

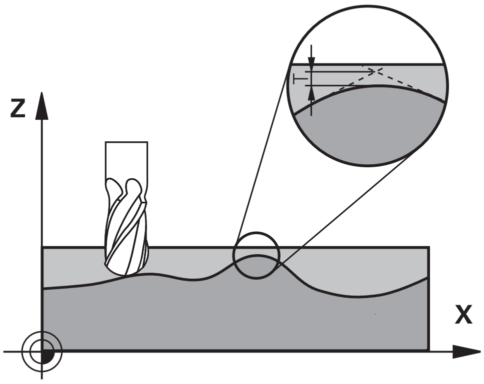
```
11 CYCL DEF 13.0 ORIENTATION
12 CYCL DEF 13.1 ANGLE180
```

19.4.3 Cycle 32 TOLERANCE

Application

Refer to your machine manual.

Machine and control must be specially prepared by the machine manufacturer for use of this cycle.



With the entries in Cycle **32** you can influence the result of HSC machining with respect to accuracy, surface definition and speed, in as much as the control has been adapted to the machine's characteristics.

The control automatically smooths the contour between any two contour elements (whether compensated or not). This means that the tool has constant contact with the workpiece surface and therefore reduces wear on the machine tool. The tolerance defined in the cycle also affects the traverse paths on circular arcs.

If necessary, the control automatically reduces the programmed feed rate so that the program can be executed at the fastest possible speed without jerking. **Even if the control does not move the axes with reduced speed, it will always comply with the tolerance that you have defined.** The larger you define the tolerance, the faster the control can move the axes.

Smoothing the contour results in a certain amount of deviation from the contour. The size of this contour error (**tolerance value**) is set in a machine parameter by the machine manufacturer. With **Cycle 32** you can change the pre-set tolerance value and select different filter settings, provided that your machine tool builder has implemented these features.

With very small tolerance values the machine cannot cut the contour without jerking. These jerking movements are not caused by poor processing power in the control, but by the fact that, in order to machine the contour transitions very exactly, the control might have to drastically reduce the speed.

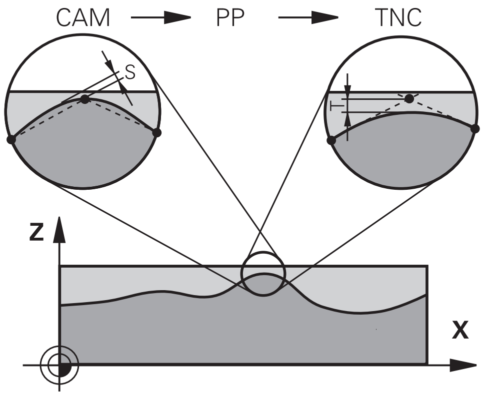
Resetting

The control resets Cycle **32** if you do one of the following:

- Redefine Cycle **32** and confirm the dialog prompt for the **tolerance value** with **NO ENT**
- Select a new NC program

After you have reset Cycle **32**, the control reactivates the tolerance that was predefined by the machine parameters.

Influences of the geometry definition in the CAM system



The most important factor of influence in offline NC program creation is the chord error S defined in the CAM system. The chord error defines the maximum point spacing of NC programs generated in a postprocessor (PP). If the chord error is less than or equal to the tolerance value **T** defined in Cycle **32**, then the control can smooth the contour points unless any special machine settings limit the programmed feed rate.

You will achieve optimal smoothing of the contour if you choose a tolerance value in Cycle **32** between 110 % and 200 % of the CAM chord error.

Notes

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- Cycle **32** is DEF-active which means that it becomes effective as soon as it is defined in the NC program.
- In a program with millimeters set as unit of measure, the control interprets the entered tolerance value **T** in millimeters. In an inch program it interprets it as inches.
- If you load an NC program with Cycle **32** that contains only the **Tolerance value** T cycle parameter, the control inserts the two remaining parameters with the value 0 if required.
- As the tolerance value increases, the diameter of circular movements usually decreases, unless HSC filters are active on your machine (set by the machine tool builder).
- If Cycle **32** is active, the control shows the defined cycle parameters on the **CYC** tab of the additional status display.

Keep the following in mind for 5-axis simultaneous machining!

- NC programs for 5-axis simultaneous machining with spherical cutters should preferably be output for the center of the sphere. The NC data are then generally more uniform. In Cycle **32**, you can additionally set a higher rotary axis tolerance **TA** (e.g., between 1° and 3°) for an even more constant feed-rate curve at the tool center point (TCP).
- For NC programs for 5-axis simultaneous machining with toroid cutters or spherical cutters, where the NC output is for the south pole of the sphere, choose a lower rotary axis tolerance. 0.1° is a typical value. However, the maximum permissible contour damage is the decisive factor for the rotary axis tolerance. This contour damage depends on the possible tool tilting, tool radius and engagement depth of the tool.
  With 5-axis hobbing with an end mill, you can calculate the maximum possible contour damage T directly from the cutter engagement length L and permissible contour tolerance TA:
  T ~ K x L x TA K = 0.0175 [1/°]
  Example: L = 10 mm, TA = 0.1°: T = 0.0175 mm

Sample formula for a toroid cutter:

When machining with a toroid cutter, the angle tolerance is very important.

$$Tw = \frac{180}{\pi * R} T_{32}$$

T_w: Angle tolerance in degrees
π: Circular constant (pi)
R: Major radius of the torus in mm
T_{32}: Machining tolerance in mm

Cycle parameters

Help graphic	Parameter
	Tolerance value T Permissible contour deviation in mm (or inches with inch programming) **> 0**: If you enter a value greater than zero, the control will use the maximum permissible deviation you have specified. **0**: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine manufacturer Input: **0...10**
	HSC mode, finishing = 0, roughing = 1 Activate filter: **0**: Milling with increased contour accuracy. The control uses internally defined finishing filter settings. **1**: Milling with increased feed rate. The control uses internally defined roughing filter settings. Input: **0**, **1**
	Tolerance TA for rotary axes Permissible position error of rotary axes in degrees with active **M128** (**FUNCTION TCPM**). The control always reduces the feed rate in such a way that—if more than one axis is traversed—the slowest axis moves at its maximum feed rate. Rotary axes are usually much slower than linear axes. You can significantly reduce the machining time for NC programs for more than one axis by entering a large tolerance value (e.g. 10°), because the control does not always have to position the rotary axis exactly at the given nominal position. The tool orientation (position of the rotary axis with respect to the workpiece surface) will be adjusted. The position at the **T**ool **C**enter **P**oint (**TCP**) will be corrected automatically. For example, with a spherical cutter measured in its center and programmed based on the center path, there will be no adverse effects on the contour. **> 0**: If you enter a value greater than zero, the control will use the maximum permissible deviation you have specified. **0**: If you enter zero or press the **NO ENT** key when programming, the control will use a value configured by the machine manufacturer. Input: **0...10**

Example

```
11 CYCL DEF 32.0 TOLERANCE
12 CYCL DEF 32.1 T0.05
13 CYCL DEF 32.2 HSC-MODE:1 TA5
```

19.5 Global Program Settings (GPS, option 44)

19.5.1 Fundamentals

Application

The Global Program Settings (GPS) allow you to define selected transformations and settings without changing the NC program. All of the settings apply globally and are superimposed on the relevant active NC program.

Related topics

- Coordinate transformations in the NC program

  Further information: "NC functions for coordinate transformation", Page 971

  Further information: "Coordinate transformation cycles", Page 958
- **GPS** tab in the **Status** workspace.

  Further information: "GPS tab (option 44)", Page 152
- Reference systems of the control

  Further information: "Reference systems", Page 934

Requirements

- Approval by the machine manufacturer

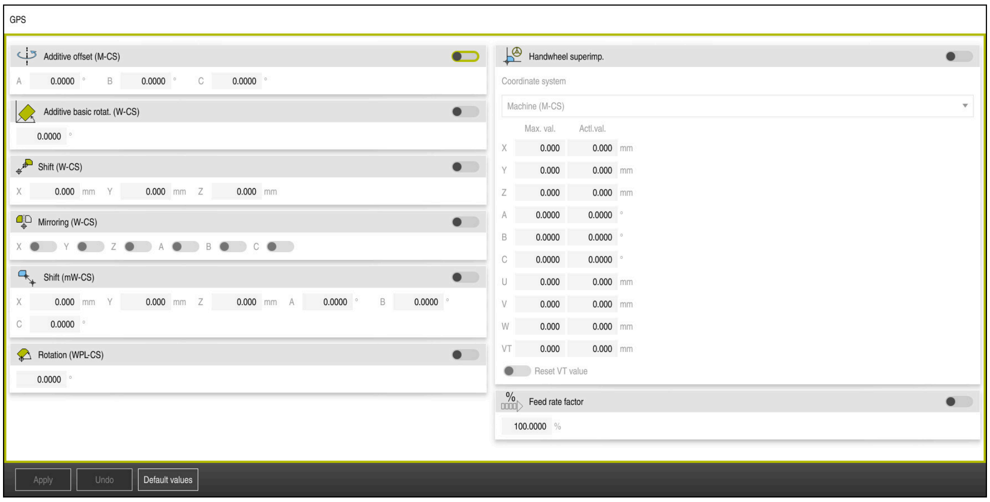
  The machine manufacturer uses the optional machine parameter **CfgGlobalSettings** (no. 128700) to define which GPS functions are available on the control.
- Global Program Settings (GPS, software option 44)

Description of function

The values of the Global Program Settings are defined and activated in the **GS** workspace.

The **GS** workspace is available in the **Program Run** operating mode and in the **MDI** application of the **Manual** operating mode.

The transformations of the **GS** workspace are effective in all operating modes and are persistent across reboots of the control.



GS workspace with active functions

The functions of GPS are activated using switches.

The control marks the sequence in which the transformations are effective with green digits.

The control shows the active settings of GPS on the **GPS** tab of the **Status** workspace.

Further information: "GPS tab (option 44)", Page 152

Before executing an NC program with active GPS in the **Program Run** operating mode, you must confirm use of the GPS functions in a pop-up window.

Buttons

The control provides the following buttons in the **GS** workspace:

Button	Description
Apply	Save changes in the **GS** workspace
Undo	Reset unsaved changes in the **GS** workspace
Default values	Set the **Feed rate factor** function to 100%, reset all other functions to zero

Overview of Global Program Settings (GPS)

The Global Program Settings (GPS) include the following functions:

Function	Description
Additive offset (M-CS)	Shift of the zero position of an axis in the machine coordinate system **M-CS** **Further information:** "Function Additive offset (M-CS)", Page 1135
Additive basic rotat. (W-CS)	Additional rotation based on basic rotation or 3D basic rotation in the workpiece coordinate system **W-CS**. **Further information:** "Function Additive basic rotat. (W-CS)", Page 1136
Shift (W-CS)	Shift of workpiece preset in a single axis in the workpiece coordinate system **W-CS** **Further information:** "Function Shift (W-CS)", Page 1137
Mirroring (W-CS)	Mirroring of individual axes in the workpiece coordinate system **W-CS** **Further information:** "Function Mirroring (W-CS)", Page 1138
Shift (mW-CS)	Additional shift of a workpiece datum already shifted in the modified workpiece coordinate system **(mW-CS)**. **Further information:** "Function Shift (mW-CS)", Page 1139
Rotation (WPL-CS)	Rotation around the active tool axis in the working plane coordinate system **WPL-CS** **Further information:** "Function Rotation (WPL-CS)", Page 1140
Handwheel superimposition	Superimposed movement of NC program positions with the electronic handwheel **Further information:** "Function Handwheel superimp.", Page 1141
Feed rate factor	Manipulation of the active feed rate **Further information:** "Function Feed rate factor", Page 1144

Define and activate Global Program Settings (GPS)

To define and activate the Global Program Settings (GPS):

- Select an operating mode, e.g. **Program run**
- Open the **GS** workspace
- Activate the switch for the required function, e.g. **Additive offset (M-CS)**
- > The control activates the selected function.
- Enter a value in the desired field, e.g. **A=10.0°**
- Apply: Press **Apply**
- > The control accepts the entered values.

> If you select an NC program for program run, you must confirm the Global Program Settings (GPS).

Resetting Global Program Settings (GPS)

To reset the Global Program Settings (GPS):

- Select an operating mode, e.g. **Program Run**
- Open the **GS** workspace

Default values

- Select **Default values**

> Provided that you have not selected the **Apply** button, you can restore the values with the **Undo** function.

> The control sets the values of all Global Program Settings (GPS) to zero except for the feed factor.
> The control sets the feed factor to 100%.

Apply

- Press **Apply**

> The control saves the values that have been reset.

Notes

- The control grays out any axes that are not active on your machine.
- Value inputs are defined in the selected unit of measurement for the position display (mm or inch), e.g. shift values and values of **Handwheel superimp.** Angles are always entered in degrees.
- The use of touch probe functions deactivates the global program settings (GPS, option 44) temporarily.

19.5.2 Function Additive offset (M-CS)

Application

With the **Additive offset (M-CS)** function, you can shift the zero position of a machine axis in the machine coordinate system **M-CS**. You can use this function, for example, on large machines, to compensate an axis when using axis angles.

Description of function

The control adds the value to the active axis-specific offset from the preset table.

Further information: "Preset table", Page 1851

If you activate a value in the **Additive offset (M-CS)** function, the zero position of the affected axis changes in the position display of the **Positions** workspace. The control assumes a different zero position of the axes.

Further information: "Positions workspace", Page 141

Application example

The travel range of a machine with AC fork head is increased using the **Additive offset (M-CS)** function. An eccentric tool chuck is used and the zero position of the C axis is shifted by 180°.

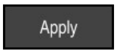
Initial situation:

- Machine kinematics with AC fork head
- Use of an eccentric tool chuck

  The tool is clamped in an eccentric tool chuck outside the center of rotation of the C axis.
- The machine parameter **presetToAlignAxis** (no. 300203) for the C axis is set to **FALSE**

To increase the traversing distance:

- Open the **GS** workspace
- Activate the **Additive offset (M-CS)** switch
- Enter **C 180°**



- Press **Apply**
- Program a positioning movement with **L C+0** in the desired NC program
- Select an NC program
- > The control considers the 180° rotation for all C axis positioning movements as well as the changed tool position.
- > The position of the C axis does not affect the position of the workpiece preset.

Notes

NOTICE

Danger of collision!

The setting of the machine parameter **presetToAlignAxis** (no. 300203) determines whether a preset is taken into account in a rotary axis offset. There is a risk of collision during the subsequent machining operation!

- Test the behavior at the machine
- If necessary, set a new preset after the offsets have been activated (mandatory for rotary axes on a table)

With the machine parameter **presetToAlignAxis** (no. 300203), the machine manufacturer defines for each axis what effect an offset of a rotary axis has on the preset:

- **True** (default): Use offset for aligning the workpiece
- **False**: Use offset for inclined-tool machining

19.5.3 Function Additive basic rotat. (W-CS)

Application

The function **Additive basic rotat. (W-CS)** facilitates e.g. better utilization of the working space. For example, you can rotate an NC program by 90° so that the X and Y directions are inverted during execution.

Description of function

The function **Additive basic rotat. (W-CS)** takes effect in addition to the basic rotation or 3D basic rotation from the preset table. The values of the preset table do not change in this respect.

Further information: "Preset table", Page 1851

The function **Additive basic rotat. (W-CS)** has no effect on the position display.

Application example

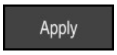
You rotate the CAM output of an NC program by 90° and compensate for the rotation using the function **Additive basic rotat. (W-CS)**.

Initial situation:

- Available CAM output for gantry-type milling machine with a large range of traverse of the Y axis
- The available machining center has the necessary traversing range only in the X axis
- The workpiece blank is clamped with a 90° rotation (long side along the X axis)
- The NC program must be rotated by 90° (algebraic sign depends on the preset position)

To rotate the CAM output:

- ▶ Open the **GS** workspace
- ▶ Activate the switch for **Additive basic rotat. (W-CS)**
- ▶ Enter **90°**



- ▶ Press **Apply**
- ▶ Select NC program
- > The control considers the 90° rotation for all axis positioning movements.

19.5.4 Function Shift (W-CS)

Application

You can use the **Shift (W-CS)** function to compensate for an offset relative to the workpiece preset for a reworking operation where probing is difficult, for example.

Description of function

The **Shift (W-CS)** function acts on an axis-by-axis basis. The value is added to an existing shift in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 940

The **Shift (W-CS)** function affects the position display. The control shifts the display by the active value.

Further information: "Position displays", Page 167

Application example

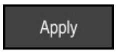
The surface of a workpiece to be reworked is determined using the handwheel and the offset is compensated for using the **Shift (W-CS)** function.

Initial situation:

- Reworking of a free-form surface is required
- Workpiece clamped
- Basic rotation and workpiece preset measured in the working plane
- Z coordinate must be defined with the handwheel due to the presence of a free-form surface

To shift the workpiece surface of a workpiece to be reworked:

- Open the **GS** workspace
- Activate the **Handwheel superimp.** switch
- Determine the workpiece surface by scratching, using the handwheel
- Activate the **Shift (W-CS)** switch
- Transfer the determined value to the corresponding axis of the **Shift (W-CS)** function, e.g. **Z**



- Press **Apply**
- Start the NC program
- Activate **Handwheel superimp.** with the **Workpiece (WPL-CS)** coordinate system
- Determine the workpiece surface by scratching, using the handwheel for fine adjustment
- Select NC program
- > The control takes the **Shift (W-CS)** into account.
- > The control uses the current values from **Handwheel superimp.** in the **Workpiece (WPL-CS)** coordinate system.

19.5.5 Function Mirroring (W-CS)

Application

You can use the **Mirroring (W-CS)** function to execute mirror-inverted execution of an NC program without having to modify the NC program.

Description of function

The **Mirroring (W-CS)** function acts on an axis-by-axis basis. The value is additive to mirroring defined in the NC program before tilting the working plane with Cycle **8 MIRRORING** or the **TRANS MIRROR** function.

Further information: "Cycle 8 MIRRORING", Page 959

Further information: "Mirroring with TRANS MIRROR", Page 973

The **Mirroring (W-CS)** function has no effect on the position display in the **Positions** workspace.

Further information: "Position displays", Page 167

Application example

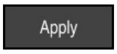
An NC program can be edited with the **Mirroring (W-CS)** function mirror-inverted.

Initial situation:

- Available CAM output for right mirror cap
- NC program set to the center of the ball-nose cutter and **FUNCTION TCPM** function with spatial angles output
- The workpiece datum is centered on the workpiece blank
- Mirroring required in the X axis to produce the left mirror cap

To mirror the CAM output of an NC program:

- Open the **GS** workspace
- Activate the **Mirroring (W-CS)** switch
- Activate the **X** switch



- Press **Apply**
- Run the NC program
- > The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.

Notes

- If you use **PLANE** functions or the **FUNCTION TCPM** function with spatial angles, the rotary axes are mirrored accordingly along with the mirrored main axes. This always creates the same constellation, regardless of whether the rotary axes were marked in the **GS** workspace.
- With **PLANE AXIAL**, the mirroring of rotary axes is irrelevant.
- With the **FUNCTION TCPM** function with axis angles, you must activate all axes to be mirrored individually in the **GS** workspace.

19.5.6 Function Shift (mW-CS)

Application

You can use the **Shift (mW-CS)** function to compensate for an offset relative to the workpiece preset for a reworking operation where probing is difficult in the modified workpiece coordinate system **mW-CS**, for example.

Description of function

The **Shift (mW-CS)** function acts on an axis-by-axis basis. The value is added to an existing shift in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 940

The **Shift (mW-CS)** function affects the position display. The control shifts the display by the active value.

"Position displays"

A modified workpiece coordinate system **mW-CS** is present with active **Shift (W-CS)** or active **Mirroring (W-CS)**. Without these preceding coordinate transformations, the **Shift (mW-CS)** option would be effective directly in the workpiece coordinate system (**W-CS**) and would thus be identical to **Shift (W-CS)**.

Application example

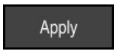
You mirror the CAM output of an NC program. After mirroring, you shift the workpiece datum in the mirrored coordinate system to produce the counterpart to a mirror cap.

Initial situation:

- Available CAM output for right mirror cap
- The workpiece datum is located in the left front corner of the workpiece blank.
- NC program set to the center of the ball-nose cutter and **Function TCPM** function with spatial angles output
- The left mirror cap is to be machined

To shift the datum in the mirrored coordinate system:

- ▶ Open the **GS** workspace
- ▶ Activate the **Mirroring (W-CS)** switch
- ▶ Activate the **X** switch
- ▶ Activate the **Shift (mW-CS)** switch
- ▶ Enter the value for shifting the workpiece datum in the mirrored coordinate system



- ▶ Press **Apply**
- ▶ Run the NC program
- > The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.
- > The control takes the modified position of the workpiece datum into account.

Notes

NOTICE

Danger of collision!

The machine parameter **presetToAlignAxis** (no. 300203) defines whether the **Shift (mW-CS)** of a rotary axis is taken into account. There is a risk of collision during the subsequent machining operation!

- ▶ Test the behavior at the machine

The **Shift (mW-CS)** function of the rotary axes to be taken into account is defined by the machine manufacturer in the machine parameter **presetToAlignAxis** (no. 300203) separately for each axis:

- **True** (default): Use offset for aligning the workpiece
- **False**: Use offset for inclined-tool machining

19.5.7 Function Rotation (WPL-CS)

Application

With the **Rotation (WPL-CS)** function, you can, for example, compensate for the misalignment of a workpiece in the already swiveled working plane coordinate system **WPL-CS** without modifying the NC program.

Description of function

The **Rotation (WPL-CS)** function is effective in the tilted working plane coordinate system **WPL-CS**. The value is additive to rotation in the NC program with Cycle **10 ROTATION** or the function **TRANS ROTATION**.

Further information: "Rotations with TRANS ROTATION", Page 975

The **Rotation (WPL-CS)** function has no effect on the position display.

19.5.8 Function Handwheel superimp.

Application

With the **Handwheel superimp.** function, you can traverse the axes with the superimposed handwheel during program run. You select the coordinate system in which the **Handwheel superimp.** function is effective.

Related topics

- Handwheel superimpositioning with **M118**

  Further information: "Activating handwheel superimpositioning with M118", Page 1239

Description of function

In the **Max. val.** column, you define the maximum traversing distance for the respective axis. You can move the input value either positively or negatively. The maximum path is therefore twice as large as the input value.

In the **Actl.val.** column, the control displays the path traversed using the handwheel for each axis.

The **Actl.val.** column can also be edited manually. If you enter a value greater than the **Max. val.**, you cannot activate the value. The control marks an incorrect value in red. The control displays a warning message and prevents the form from being closed.

If the **Actl.val.** column contains a value when you activate the function, the control will use the menu for returning to move to the new position.

Further information: "Returning to the contour", Page 1793

The **Handwheel superimp.** function affects the position display in the **Positions** workspace. The control shows the values offset by the handwheel in the position display.

Further information: "Positions workspace", Page 141

The control displays the values of the two methods for **Handwheel superimp.** on the **POS HR** tab of the additional status display.

On the **POS HR** tab of the **Status** workspace, the control shows whether the **Max. val.** is defined using the **M118** function or the Global Program Settings (GPS).

Further information: "POS HR tab", Page 158

Virtual tool axis VT

The virtual tool axis **VT** is needed for machining operations with inclined tools, e.g. for manufacturing oblique holes without using a tilted working plane.

Handwheel superimp. can also be executed in the active tool axis direction. The **VT** always corresponds to the direction of the active tool axis. On machines with head rotation axes, this direction may not correspond to the basic coordinate system **B-CS**. You activate the function with the **VT** line.

Further information: "Notes concerning different machine kinematics", Page 979

By default, values traversed with the handwheel in the **VT** remain active even after a tool change. If you activate the **Reset VT value** switch, the control resets the actual value of the **VT** when a tool is changed.

The control displays the values of the virtual tool axis **VT** on the **POS HR** tab of the **Status** workspace.

Further information: "POS HR tab", Page 158

For the control to display values, you must define a value greater than 0 in the **VT** function for **Handwheel superimp.**

Notes

NOTICE

Danger of collision!

The coordinate system chosen in the selection menu also takes effect on **Handwheel superimp.** with **M118**, even if the Global Program Settings function (GPS) is not active. There is a risk of collision during the execution of **Handwheel superimp.** and the subsequent machining operations!

- Before exiting the form, always make sure to select the **Machine (M-CS)** coordinate system
- Test the behavior at the machine

NOTICE

Danger of collision!

When both methods for **Handwheel superimp.** with **M118** and with the Global Program Settings GPS are effective at the same time, the definitions influence each other, depending on their sequence of activation. There is a risk of collision during the execution of **Handwheel superimp.** and the subsequent machining operations!

- Use only one method for **Handwheel superimp.**
- Preferably use the **Handwheel superimp.** option of the **Global Program Settings** function
- Test the behavior at the machine

HEIDENHAIN does not recommend using both methods for **Handwheel superimp.** at the same time. If **M118** cannot be removed from the NC program, you should at least activate **Handwheel superimp.** from GPS prior to selecting the program. This ensures that the control uses the GPS function rather than **M118**.

- If neither the NC program nor the Global Program Settings were used to activate coordinate system transformations, **Handwheel superimp.** is effective in the same manner in all coordinate systems.
- If you want to use **Handwheel superimp.** while machining with active Dynamic Collision Monitoring (DCM), then the control must be in a stopped or interrupted state. Alternatively, you can also deactivate DCM.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084
- **Handwheel superimp.** in virtual axis direction **VT** requires neither a **PLANE** function nor the **FUNCTION TCPM** function.
- Use the machine parameter **axisDisplay** (no. 100810) to define whether the control also shows the virtual axis **VT** in the position display of the **Positions** workspace.

  Further information: "Positions workspace", Page 141

19.5.9 Function Feed rate factor

Application

You can use the **Feed rate factor** function to influence the effective feed rates on the machine, e.g. to adjust the feed rates of a CAM program. This will prevent the CAM program from being re-output using the postprocessor. When doing so, you change all feed rates as a percentage without making any changes in the NC program.

Related topics

- Feed rate limit **F MAX**

  The **Feed rate factor** function has no influence on the feed rate limit with **F MAX**.

  Further information: "Feed rate limit F MAX", Page 1781

Description of function

All feed rates are changed as a percentage. You define a percentage value from 1% to 1000%.

The **Feed rate factor** function acts on the programmed feed rate and the feed rate potentiometer, but not on rapid traverse **FMAX**.

The control shows the current feed rate in field **F** of the **Positions** workspace. If the **Feed rate factor** function is active, the feed rate is shown with the defined values taken into account.

Further information: "Presets and technology values", Page 143

20

Monitoring

20.1 Component Monitoring with MONITORING HEATMAP (option 155)

Application

The **MONITORING HEATMAP** function allows you to start and stop the workpiece representation in a component heatmap from within the NC program.

The control monitors the selected component and shows the result in a heatmap on the workpiece.

> If the process monitoring (option 168) in the simulation displays a process heat map, the control does not display a component heat map.
>
> **Further information:** "Process Monitoring (option 168)", Page 1152

Related topics

- **MON** tab in the **Status** workspace

  Further information: "MON tab (option 155)", Page 155
- Cycle **238 MEASURE MACHINE STATUS** (option 155)

  Further information: "Cycle 238 MEASURE MACHINE STATUS (option 155)", Page 1150
- Color the workpiece as a heat map in the simulation

  Further information: "Workpiece options column", Page 1409
- **Process Monitoring** (option 168) with **SECTION MONITORING**

  Further information: "Process Monitoring (option 168)", Page 1152

Requirements

- Component Monitoring (software option 155)
- Components to be monitored are defined

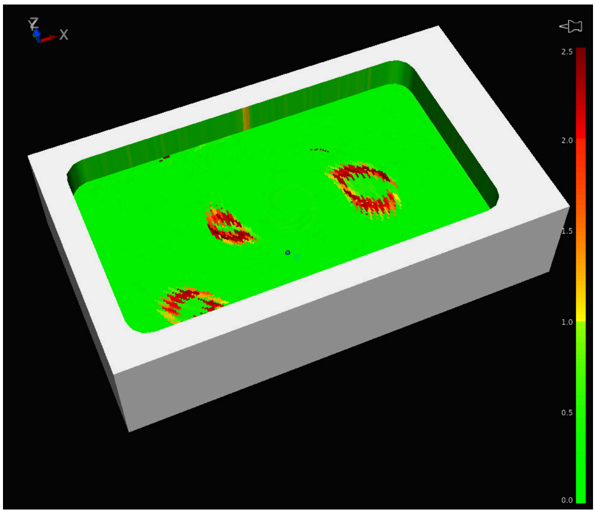
  In the optional machine parameter **CfgMonComponent** (no. 130900), the machine manufacturer defines the machine components to be monitored as well as the warning and error thresholds.

Description of function

A component heatmap is similar to the image from an infrared camera.

- Green: component works under conditions defined as safe
- Yellow: component works under warning zone conditions
- Red: Overload condition

The control shows these statuses on the workpiece in the simulation and can overwrite the statuses upon subsequent operations.


Representation of the component heat map in the simulation with missing pre-machining

Only one component at a time can be monitored with the heatmap. If you start the heatmap several times in a row, monitoring of the previous component is stopped.

Input

11 MONITORING HEATMAP START FOR "Spindle"	; Activate monitoring of the **Spindle** component and display it as a heat map

The NC function includes the following syntax elements:

Syntax element	Meaning
MONITORING HEATMAP	Syntax initiator for component monitoring
START FOR or **STOP**	Start or stop component monitoring
" " or **QS**	Fixed or variable name of the component to be monitored Only if **START FOR** is selected

Note

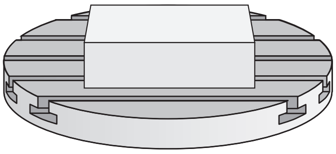
The control cannot display changes in the statuses directly in the simulation, as it must process the incoming signals, e.g. in the event of a tool breakage. The control shows the change with a slight time delay.

20.2 Cycles for monitoring

20.2.1 Cycle 239 ASCERTAIN THE LOAD (option 143)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



The dynamic behavior of your machine may vary with different workpiece weights acting on the machine table. A change in the load has an influence on the friction forces, acceleration, holding torque and stick-slip friction of the table axes. With option 143 LAC (Load Adaptive Control) and Cycle **239 ASCERTAIN THE LOAD**, the control is able to automatically determine and adjust the actual mass inertia of the load, the actual friction forces, and the maximum axis acceleration or reset the feedforward and controller parameters. In this way, you can optimally react to major load changes. The control performs a weighing procedure to ascertain the weight acting on the axes. With this weighing run, the axes move by a specified distance. Your machine tool builder defines the specific movements. Before weighing, the axes are moved to a position, if required, where there is no danger of collision during the weighing procedure. This safe position is defined by the machine tool builder.

In addition to adjusting the control parameters, with LAC the maximum acceleration is also adjusted in accordance with the weight. This enables the dynamics to be accordingly increased with low load to increase productivity.

Cycle sequence

Parameter Q570 = 0

1 There is no physical movement of the axes.
2 The control resets the LAC.
3 The control activates feedforward and, if applicable, controller parameters that allow safe movements of the axis/axes, independently of the current load condition. The parameters set with **Q570**=0 are **independent** of the current load
4 These parameters can be useful during the setup procedure or after the completion of an NC program.

Parameter Q570 = 1

1 The control performs a weighing procedure in which it moves one or more axes. Which axes are moved depends on the setup of the machine and on the drives of the axes.
2 The scope of axis movement is defined by the machine tool builder.
3 The feedforward and controller parameters determined by the control **depend** on the current load.
4 The control activates the ascertained parameters.

If you are using the mid-program startup function and the control thus skips Cycle **239** in the block scan, the control will ignore this cycle—no weighing run will be performed.

Notes

NOTICE

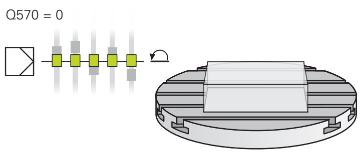
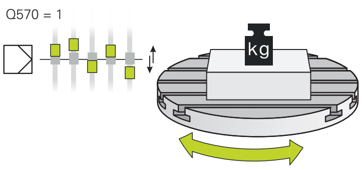
Danger of collision!

The cycle can execute extensive movements in several axes at rapid traverse!

- Contact your machine tool builder to learn about the type and range of movements in Cycle **239** before using the cycle.
- Before the cycle starts, the control moves to a safe position, if applicable. The machine tool builder determines this position.
- Set the potentiometers for feed-rate and rapid-traverse override to at least 50 % to ensure a correct ascertainment of the load.

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- Cycle **239** becomes effective immediately after its definition.
- Cycle **239** supports the determination of the load on synchronized axes (gantry axes) if they have only one common position encoder (torque master slave).

Cycle parameters

Help graphic	Parameter
 	**Q570 Load (0 = Delete/1 = Ascertain)?** Define whether the control will perform a LAC (Load Adaptive Control) weighing run, or whether the most recently ascertained load-dependent feedforward and controller parameters will be reset: **0**: Reset LAC; the values most recently ascertained by the control are reset, and the control uses load-independent feedforward and controller parameters **1**: Perform a weighing run; the control moves the axes and thus ascertains the feedforward and controller parameters depending on the current load. The values ascertained are activated immediately. Input: **0**, **1**

Example

```
11 CYCL DEF 239 ASCERTAIN THE LOAD ~
    Q570=+0                        ;LOAD ASCERTATION
```

20.2.2 Cycle 238 MEASURE MACHINE STATUS (option 155)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.

During their lifecycle, the machine components which are subject to loads (e.g. guides, ball screws, ...) become worn and thus, the quality of the axis movements deteriorates. This, in turn, affects the production quality.

Using **Component Monitoring** (Option 155) and Cycle **238**, the control is able to measure the current machine status. As a result, any deviations from the machine's shipping condition due to wear and aging can be measured. The measurement results are stored in a text file that is readable for the machine tool builder. He can read and evaluate the data, and react with predictive maintenance, thereby avoiding unplanned machine downtimes.

The machine tool builder can define warning and error thresholds for the measured values and optionally specify error reactions.

Cycle sequence

Ensure that the axes are not clamped before you start the measurement.

Parameter Q570=0

1 The control performs movements in the machine axes
2 The feed rate, rapid traverse, and spindle potentiometers are effective

Your machine tool builder defines in detail how the axes will move.

Parameter Q570=1

1 The control performs movements in the machine axes
2 The feed rate, rapid traverse, and spindle potentiometers are **not** effective
3 On the **MON** status tab, you can select the monitoring task to be displayed
4 This diagram allows you to watch how close the components are to a warning or error threshold

Further information: "MON tab (option 155)", Page 155

Your machine tool builder defines in detail how the axes will move.

Notes

NOTICE

Danger of collision!

This cycle may perform extensive movements in one or more axes at rapid traverse! If you program the cycle parameter **Q570** = 1, the feed rate and rapid traverse potentiometers, and, if applicable, the spindle potentiometer, have no effect. However, you can stop any movement by setting the feed rate potentiometer to zero. There is a danger of collision!

- Before recording measured data, test the cycle in test mode with **Q570 = 0**
- Contact your machine manufacturer to learn about the type and range of movements in Cycle **238** before using the cycle.

- This cycle can be executed in the **FUNCTION MODE MILL**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** machining modes.
- Cycle **238** is CALL-active.
- If, during a measurement, you set, for example, the feed rate potentiometer to zero, then the control will abort the cycle and display a warning. You can acknowledge the warning by pressing the **CE** key and then press the **NC start** key to run the cycle again.

Cycle parameters

Help graphic	Parameter
	Q570 Mode (0=test/1=measure)? Define whether the control will perform a measurement of the machine status in test mode or in measurement mode: **0**: No measured data will be generated. You can control the axis movements with the feed rate and rapid traverse potentiometers **1**: This mode will generate measured data. You **cannot** control the axis movements with the feed rate and rapid traverse potentiometers Input: **0**, **1**

Example

```
11 CYCL DEF 238  MEASURE MACHINE STATUS ~
    Q570=+0                         ;MODE
```

20.3 Process Monitoring (option 168)

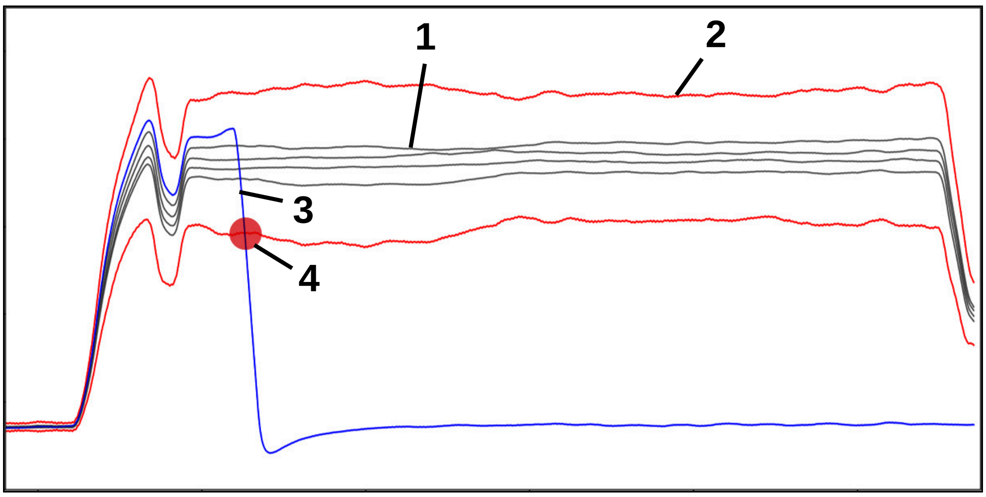
20.3.1 Fundamentals

The control uses process monitoring to detect disturbances in the machining process, e.g:.

- Tool breakage
- Incorrect or missing workpiece pre-machining
- Changed position or size of the workpiece blank
- Wrong material, e.g. aluminum instead of steel

Process monitoring allows you to monitor the machining process during program run using monitoring tasks. The monitoring task compares the signal curve of the current execution of an NC program with one or more reference machining operations. The monitoring task uses these reference machining operations to determine an upper and lower limit. If the current machining operation is outside the limits for a defined hold time, the monitoring task responds with a defined reaction. If, for example, the spindle current drops due to a tool breakage, the monitoring task stops the NC program.

Further information: "Interrupting, stopping or canceling program run", Page 1782



Drop in spindle current due to tool breakage

1	References
2	Limits consisting of tunnel width and, if necessary, expansion
3	Current machining operation
4	Process fault, e.g. due to tool breakage

If you are using process monitoring, the following steps are required:

- Defining monitoring sections in the NC program

  Further information: "Defining monitoring sections with MONITORING SECTION (option 168)", Page 1175
- Slowly running-in the NC program in Single Block mode before activating process monitoring

  Further information: "Program Run", Page 1777
- Activating process monitoring

  Further information: "Monitoring options column", Page 1169
- If necessary, configuring settings for the monitoring tasks
  - Selecting a strategy template

    Further information: "Strategy template", Page 1159
  - Adding or removing monitoring tasks

    Further information: "Icons", Page 1155
  - Defining settings and responses within the monitoring tasks

    Further information: "Monitoring tasks", Page 1161
  - Displaying monitoring task in the simulation as a process heat map

    Further information: "Monitoring options column within a monitoring section", Page 1170
    Further information: "Workpiece options column", Page 1409
- Running the NC program in Full Sequence

  Further information: "Program Run", Page 1777
- Selecting the required references as necessary for the monitoring tasks

  Further information: "Monitoring tasks", Page 1161

  Further information: "Records of monitoring sections", Page 1172

Related topics

- **Component monitoring** (option 155) with **MONITORING HEATMAP**

  Further information: "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1146

20.3.2 Process Monitoring workspace (option 168)

Application

In the **Process Monitoring** workspace the control visualizes the machining process during program run. You can activate various monitoring tasks that are relevant to the process. If necessary, you can adapt the monitoring tasks.

Further information: "Monitoring tasks", Page 1161

Requirements

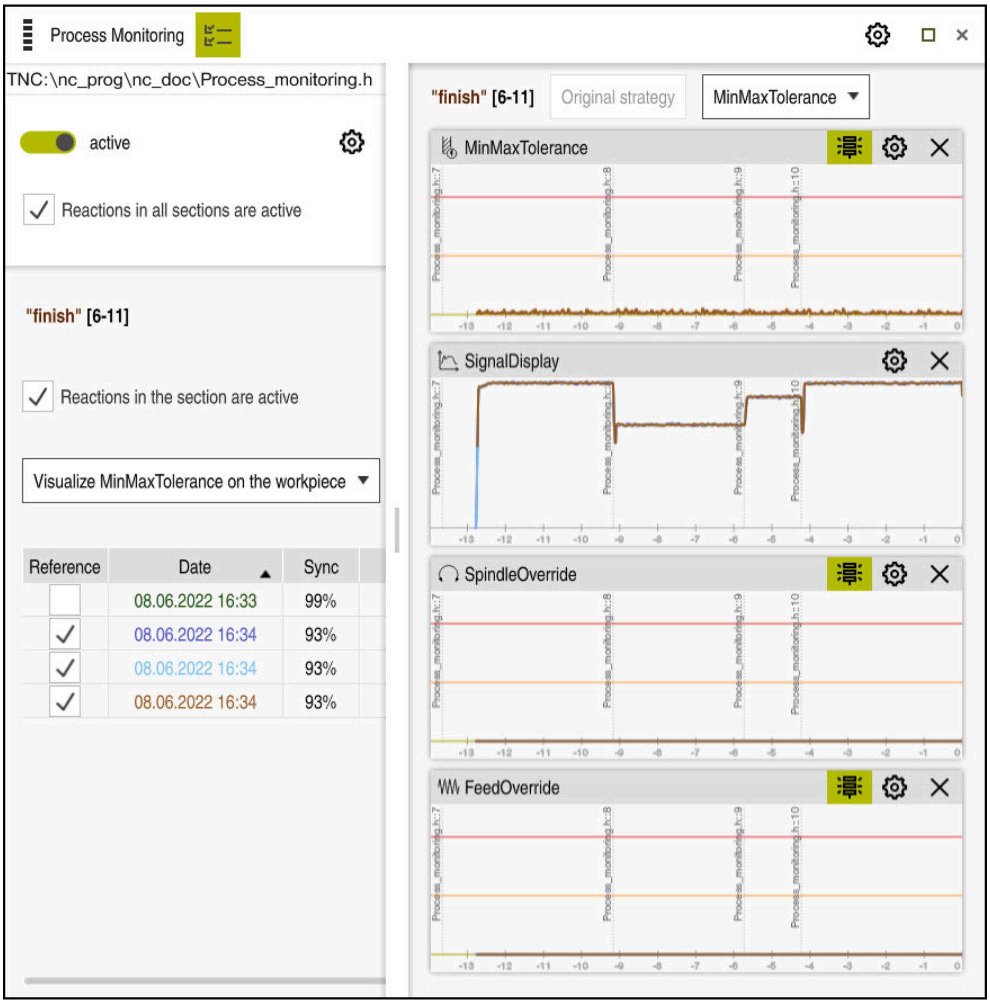
- Process Monitoring (software option 168)
- Monitoring sections defined with **MONITORING SECTION**

  Further information: "Defining monitoring sections with MONITORING SECTION (option 168)", Page 1175
- Possible without limitation in the **FUNCTION MODE MILL** machining mode

  The monitoring tasks **FeedOverride** and **SpindleOverride** are functional in the **FUNCTION MODE TURN** machining mode (option 50).

Functionality

The **Process Monitoring** workspace provides information and settings for monitoring the machining process.



Process Monitoring workspace

Depending on the cursor position in the NC program, the control provides the following areas:

- Global area

  The control shows information about the active NC program.

  Further information: "Global area", Page 1156
- Strategy range

  The control shows the monitoring tasks and the graphs of the recordings. You can configure settings for the monitoring tasks.

  Further information: "Strategy range", Page 1158
- **Monitoring options** column in the global range

  The control displays information on the recordings that relate to all monitoring sections of the NC program.

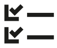
  Further information: "Monitoring options column in the global range", Page 1170
- **Monitoring options** column within a monitoring section

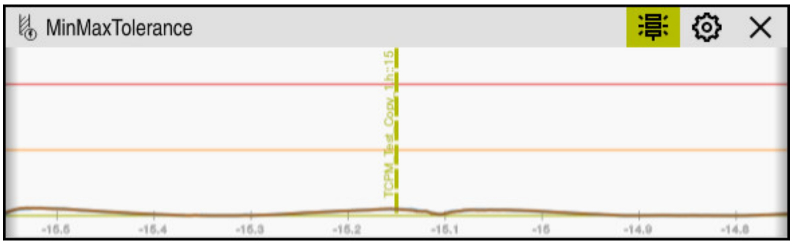
  The control displays information on the recordings that relate only to the currently selected monitoring section.

  Further information: "Monitoring options column within a monitoring section", Page 1170

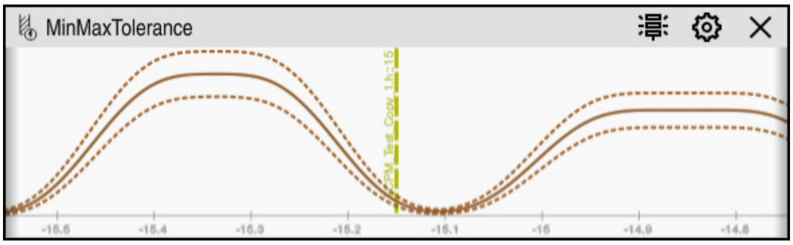
Icons

The following icons are shown in the **Process Monitoring** workspace:

Icon	Meaning
	Show or hide the **Monitoring options** column **Further information:** "Monitoring options column", Page 1169
	Remove monitoring task **Further information:** "Monitoring tasks", Page 1161
	Add monitoring task **Further information:** "Monitoring tasks", Page 1161
	Open settings You can open the following settings: ■ **Process Monitoring** workspace setting **Further information:** "Settings for the Process Monitoring workspace", Page 1168 ■ Setting in the **Settings for NC program** window of the **Monitoring options** column **Further information:** "Settings for NC program window", Page 1174 ■ Monitoring task setting **Further information:** "Monitoring tasks", Page 1161
	Show or hide warning and error limits If you show the warning and error limits, the control shows the monitored signal in relation to the defined limits. The control shows the following warning and error limits: ■ Green line If the current machining operation is at the bottom line, the current machining operation corresponds to the reference. ■ Orange line This line shows the warning limit. If the current machining operation exceeds the middle line, the current machining operation deviates by half the set limit of the reference. ■ Red line This line shows the error limit. If the current machining operation exceeds the upper line for a defined hold time, the monitoring task triggers a defined response, e.g. NC stop. If you hide the warning and error limits, the control shows an absolute display of the monitored signal. The dashed lines represent the upper and lower error limits, i.e. the tunnel width.



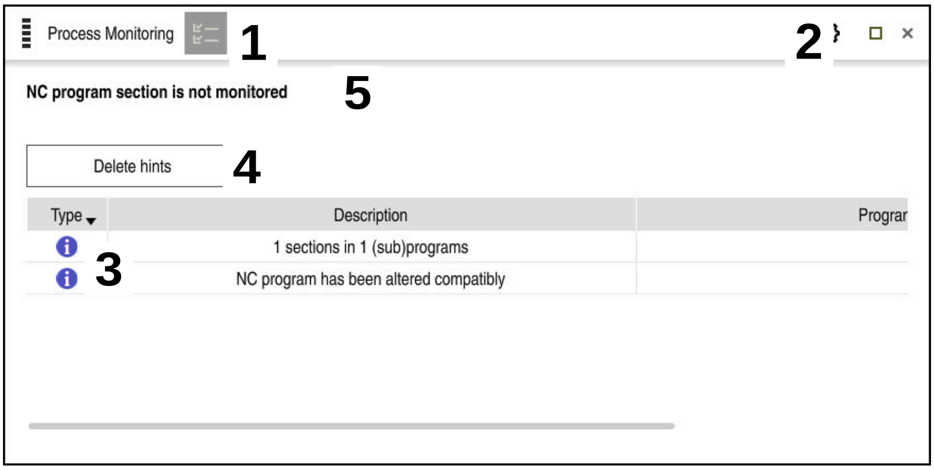
Warning and error limits displayed: The control shows the signal in relation to the defined limits



Warning and error limits hidden: The solid line represents the signal and the dashed lines represent the tunnel width determined at the time

Global area

If the cursor is outside a monitoring section in the NC program, the **Process Monitoring** workspace displays the global range.



Global area in the **Process Monitoring** workspace

The **Process Monitoring** workspace shows the following in the global range:

1 **Monitoring options** icon
   Further information: "Monitoring options column", Page 1169
2 **Settings** icon for the **Process Monitoring** workspace
   Further information: "Settings for the Process Monitoring workspace", Page 1168
3 Table with notes on the active NC program
   Further information: "Notes on the NC program", Page 1157
4 **Delete hints** button
   You can use the **Delete hints** button to empty the table.
5 Information that this area is not monitored in the NC program

Notes on the NC program

In this area, the control shows a table with information about the active NC program. The table contains the following information:

Column or symbol	Meaning
Type	In the **Type** column, the control shows different types of notifications.
	Information, e.g. the number of monitoring sections
	Warning, e.g. if a monitoring section has been removed
	Error, e.g. if you should reset the recordings If you make changes within a monitoring section, that monitoring section can no longer be monitored. Therefore, you should reset the recordings and set new references so that machining is monitored again. **Further information:** "Settings for NC program window", Page 1174 You can sort the table by information type by selecting the **Type** column.
Description	In the **Description** column, the control displays information about the information types, e.g.: ■ Changes to the NC program ■ Cycles contained in the NC program ■ Interruptions, e.g. **M0** or **M1**
Program line	If the information depends on an NC block number, the control displays the program name and the NC block number.

Strategy range

If the cursor is inside a monitoring section in the NC program, the **Process Monitoring** workspace displays the strategy range.

Strategy range in the **Process Monitoring** workspace

The **Process Monitoring** workspace shows the following in the strategy range:

1 **Monitoring options** icon
Further information: "Monitoring options column", Page 1169
2 **Settings** icon for the **Process Monitoring** workspace
Further information: "Settings for the Process Monitoring workspace", Page 1168
3 **Settings** icon for the monitoring tasks
Further information: "Monitoring tasks", Page 1161
4 Show or hide warning and error limits
Further information: "Icons", Page 1155
5 Monitoring tasks
Further information: "Monitoring tasks", Page 1161

6 The control shows the following information and functions:
   - Name of the monitoring section, if applicable
     If **AS** is defined in the NC program with the optional syntax element, the control displays the name.
     If no name is defined, the control displays **MONITORING SECTION**.
     Further information: "Input", Page 1175
   - Range of NC block numbers of the monitoring section in square brackets
     Start and end of the monitoring section in the NC program
   - **Original strategy** or **Save strategy as template** button
     Further information: "Strategy template", Page 1159
   - Selection menu for strategy template
     Further information: "Strategy template", Page 1159

Strategy template

A strategy template includes one or more monitoring tasks, including the defined settings.

You can choose between the following strategy templates using a selection menu:

Strategy template	Meaning
MinMaxTolerance	This strategy template includes the following monitoring tasks: ■ **MinMaxTolerance** **Further information:** "Monitoring task MinMaxTolerance", Page 1162 ■ **SignalDisplay** **Further information:** "Monitoring task SignalDisplay", Page 1166 ■ **SpindleOverride** **Further information:** "Monitoring task SpindleOverride", Page 1166 ■ **FeedOverride** **Further information:** "Monitoring task FeedOverride", Page 1167
StandardDeviation	This strategy template includes the following monitoring tasks: ■ **StandardDeviation** **Further information:** "Monitoring task StandardDeviation", Page 1165 ■ **SignalDisplay** **Further information:** "Monitoring task SignalDisplay", Page 1166 ■ **SpindleOverride** **Further information:** "Monitoring task SpindleOverride", Page 1166 ■ **FeedOverride** **Further information:** "Monitoring task FeedOverride", Page 1167
User-defined	In this strategy template, you can compile the monitoring tasks yourself.

If you modify a strategy template, you can overwrite the modified strategy template by clicking the **Save strategy as template** button. The control overwrites the currently selected strategy template.

> Since you cannot restore the as-delivered state of the strategy templates yourself, only overwrite the **User-defined** template.
>
> The machine manufacturer can use the optional machine parameter **ProcessMonitoring** (no. 133700) to restore the as-delivered state of the strategy templates.

In the settings of the **Process Monitoring** workspace, you can define which strategy template the control selects by default after creating a new monitoring section.

Further information: "Settings for the Process Monitoring workspace", Page 1168

Monitoring tasks

You can change the settings and responses of the monitoring tasks for each monitoring section. You can also remove a monitoring task for a monitoring section or add one using the plus sign.

The **Process Monitoring** workspace contains the following monitoring tasks:

- **MinMaxTolerance**

  With **MinMaxTolerance**, the control monitors whether the current machining operation is within the range of the selected references including the percentage deviation.

  Further information: "Monitoring task MinMaxTolerance", Page 1162
- **StandardDeviation**

  With **StandardDeviation**, the control monitors whether the current machining operation is within the range of the selected references, including expansion by the factor σ.

  Further information: "Monitoring task StandardDeviation", Page 1165
- **SignalDisplay**

  With **SignalDisplay**, the control shows the process progress of all selected references and the current machining operation.

  Further information: "Monitoring task SignalDisplay", Page 1166
- **SpindleOverride**

  With **SpindleOverride**, the control monitors changes to the spindle override by the potentiometer.

  Further information: "Monitoring task SpindleOverride", Page 1166
- **FeedOverride**

  With **FeedOverride**, the control monitors changes in the feed override by the potentiometer.

  Further information: "Monitoring task FeedOverride", Page 1167

The values of the monitoring tasks set in the as-delivered state are considered recommended output values, which you can adapt if necessary in special machining situations.

In each monitoring task, the control shows the current processing and the selected references as a graph. The time axis is specified in seconds.

Monitoring task MinMaxTolerance

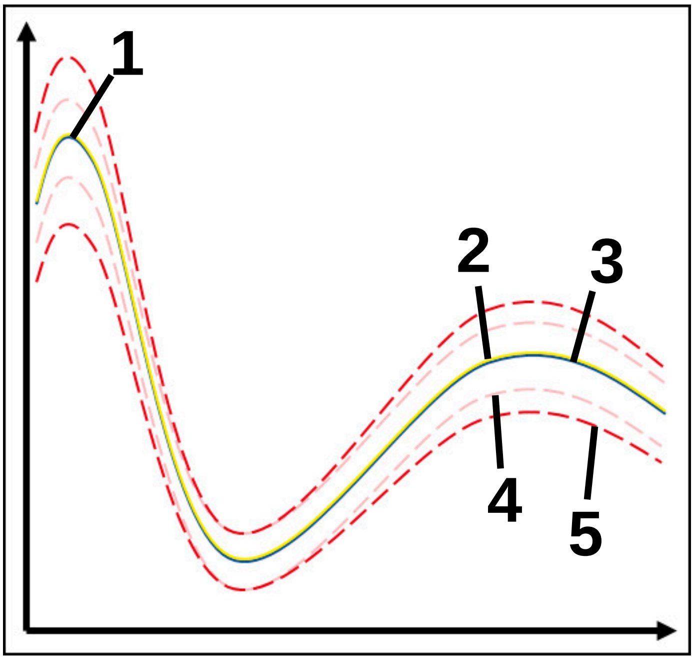
With **MinMaxTolerance**, the control monitors whether the current machining operation is within the range of the selected references including the percentage deviation.

The percentage deviation takes into account the tool wear.

The application cases of **MinMaxTolerance** are significant process faults, e.g. during small series production:

- Tool breakage
- Missing tool
- Changed position or size of the workpiece blank

The control requires at least one recorded machining operation as a reference. If you do not select a reference, this monitoring task is inactive and does not draw a graph.



1	—	First good reference
2	—	Second good reference
3	—	Third good reference
4	– –	Limits consisting of the tunnel width
5	– –	Limits consisting of a percentage expansion of the tunnel width

Further information: "Records of monitoring sections", Page 1172

If, for example, you have a recording that is only just acceptable due to tool wear, you can also use an alternative application with this monitoring task.

Further information: "Alternative application with acceptable reference", Page 1164

Settings for MinMaxTolerance

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**

  Percentage expansion of tunnel width
- **Static tunnel width**

  Upper and lower limits, based on references
- **Hold time**

  Maximum time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control triggers the defined response of the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Issue warning message**

  If the signal exceeds the limits for the defined hold time, the control shows a warning in the notification menu.

  Further information: "Message menu on the information bar", Page 1400
- **Stop NC program**

  If the signal exceeds the warning limits for the defined hold time, the control stops the NC program.
- **Lock current tool**

  If the signal exceeds the warning limits for the defined hold time, the control blocks the tool in the tool management.

  Further information: "Tool management ", Page 270

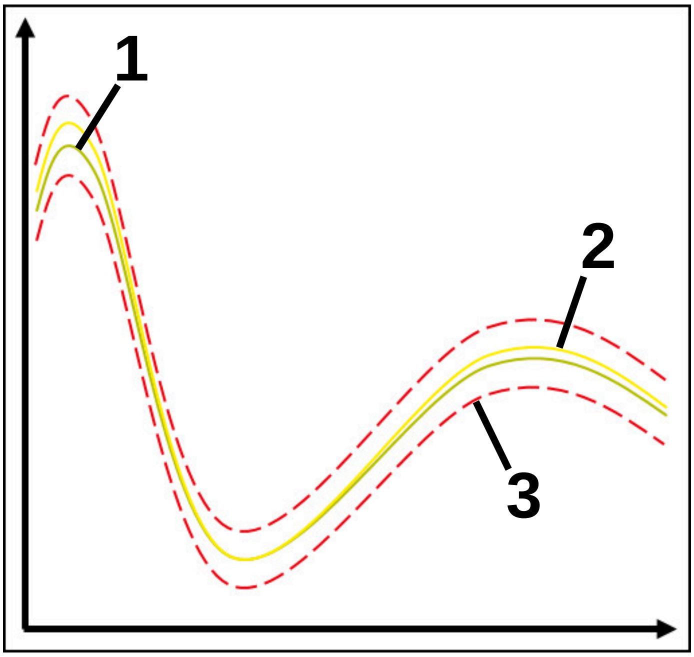
Alternative application with acceptable reference

If the control has recorded a machining operation that is only just acceptable, you can use an alternative application of the monitoring task **MinMaxTolerance**.

Select at least two references:

- An optimal reference
- A reference that is only just acceptable, e.g. showing a higher signal of the spindle load due to tool wear

The monitoring task checks whether the current machining operation is within the range of the selected references. For this strategy, select no deviation or a low percentage deviation, since the tolerance is already given by the different references.



1	▬	Optimal reference
2	▬	Reference only just acceptable
3	‒ ‒	Limits consisting of the tunnel width

Monitoring task StandardDeviation

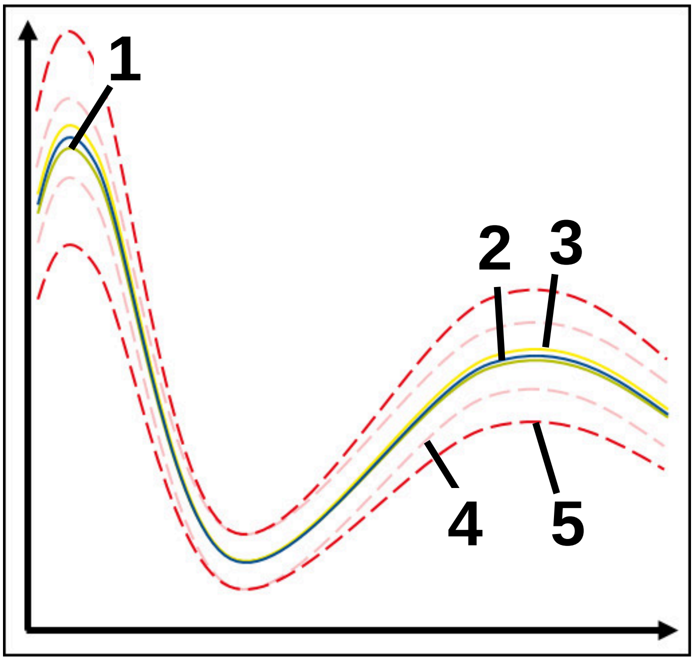
With **StandardDeviation**, the control monitors whether the current machining operation is within the range of the selected references, including expansion by the factor σ.

The application cases of **StandardDeviation** are process faults of all kinds, e.g. during series production:

- Tool breakage
- Missing tool
- Tool wear
- Changed position or size of the workpiece blank

The control requires at least three recorded machining operations as a reference. The references should include one optimal, one good and one acceptable machining operation. If you do not select the required references, this monitoring task is not active and does not draw a graph.

Further information: "Records of monitoring sections", Page 1172



1		Optimal reference
2		Good reference
3		Reference only just acceptable
4		Limits consisting of the tunnel width
5		Limits consisting of the expansion of the tunnel width multiplied by factor σ

Settings for StandardDeviation

You can use sliders to configure the following settings for this monitoring task:

- **A multiple of σ**

  Expansion of the tunnel width multiplied by factor σ
- **Static tunnel width**

  Upper and lower limits, based on references
- **Hold time**

  Maximum time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control triggers the defined response of the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Issue warning message**

  If the signal exceeds the limits for the defined hold time, the control shows a warning in the notification menu.

  Further information: "Message menu on the information bar", Page 1400
- **Stop NC program**

  If the signal exceeds the warning limits for the defined hold time, the control stops the NC program.
- **Lock current tool**

  If the signal exceeds the warning limits for the defined hold time, the control blocks the tool in the tool management.

  Further information: "Tool management ", Page 270

Monitoring task SignalDisplay

With **SignalDisplay**, the control shows the process progress of all selected references and the current machining operation.

You can compare whether the current machining operation corresponds to the references. This allows you to visually check whether you can use the machining operation as a reference.

The monitoring task does not respond.

Monitoring task SpindleOverride

With **SpindleOverride**, the control monitors changes to the spindle override by the potentiometer.

The control uses the first recorded machining operation as a reference.

Settings for SpindleOverride

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**

  Accepted deviation of the override in percent compared to the first recording
- **Hold time**

  Maximum time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control triggers the defined response of the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Issue warning message**

  If the signal exceeds the limits for the defined hold time, the control shows a warning in the notification menu.

  Further information: "Message menu on the information bar", Page 1400
- **Stop NC program**

  If the signal exceeds the warning limits for the defined hold time, the control stops the NC program.

Monitoring task FeedOverride

With **FeedOverride**, the control monitors changes in the feed override by the potentiometer.

The control uses the first recorded machining operation as a reference.

FeedOverride settings

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**

  Accepted deviation of the override in percent compared to the first recording
- **Hold time**

  Maximum time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control triggers the defined response of the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

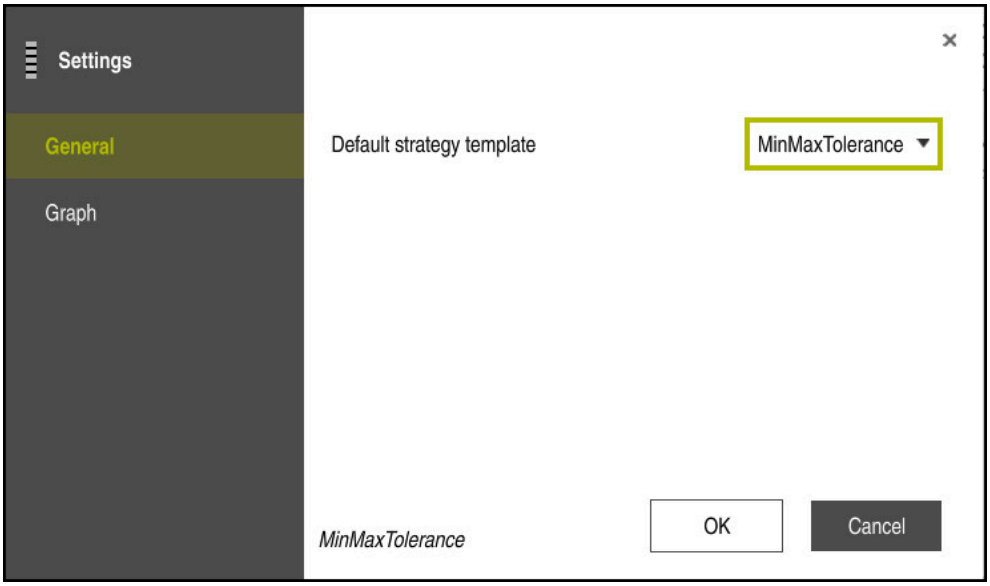
- **Issue warning message**

  If the signal exceeds the limits for the defined hold time, the control shows a warning in the notification menu.

  Further information: "Message menu on the information bar", Page 1400
- **Stop NC program**

  If the signal exceeds the warning limits for the defined hold time, the control stops the NC program.

Settings for the Process Monitoring workspace



Settings for the **Process Monitoring** workspace

General

In the **General** area, select which strategy template the control uses as the default:

- **MinMaxTolerance**
- **StandardDeviation**
- **User-defined**

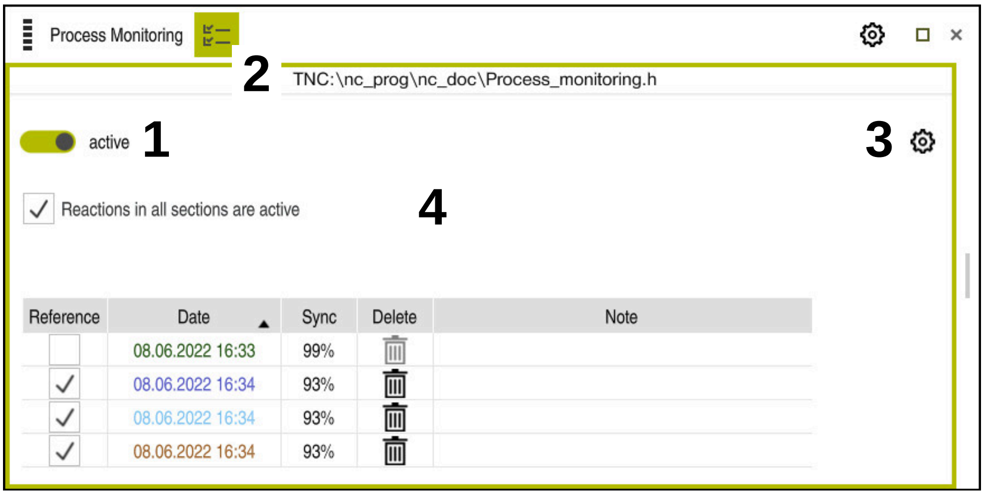
Further information: "Strategy template", Page 1159

Graph

In the **Graph** area, you can select the following settings:

Setting	Meaning
Simultaneously plotted references	Select the maximum number of recordings that the control displays simultaneously as graphs in the monitoring tasks: ■ **2** ■ **4** ■ **6** ■ **8** ■ **10** If more references are selected than the control is to display, the control displays the last selected references as a graph.
Preview [s]	The control can run a selected reference as a preview during processing. The control shifts the time axis of the machining operation to the left. Select how many seconds of the reference the control will preview: ■ **0** ■ **2** ■ **4** ■ **6** **Further information:** "Records of monitoring sections", Page 1172

Monitoring options column



Monitoring options column in the global range

The **Monitoring options** column shows the following in the upper area regardless of the cursor position in the NC program:

1 Switch for activating or deactivating process monitoring for the entire NC program
2 Path of the current NC program
3 Open the **Settings** icon in the **Settings for NC program** window
Further information: "Settings for NC program window", Page 1174
4 Checkbox for activating or deactivating the responses of all monitoring sections in the NC program

Depending on the cursor position in the NC program, the control provides the following areas:

- **Monitoring options** column in the global range
  You can select references that are effective for all monitoring sections of the NC program.
  Further information: "Monitoring options column in the global range", Page 1170
- **Monitoring options** column within a monitoring section
  You can define settings and select references that apply to the currently selected monitoring section.
  Further information: "Monitoring options column within a monitoring section", Page 1170

Monitoring options column in the global range

If the cursor is outside a monitoring section in the NC program, the **Process Monitoring** workspace displays the **Monitoring options** column in the global area.

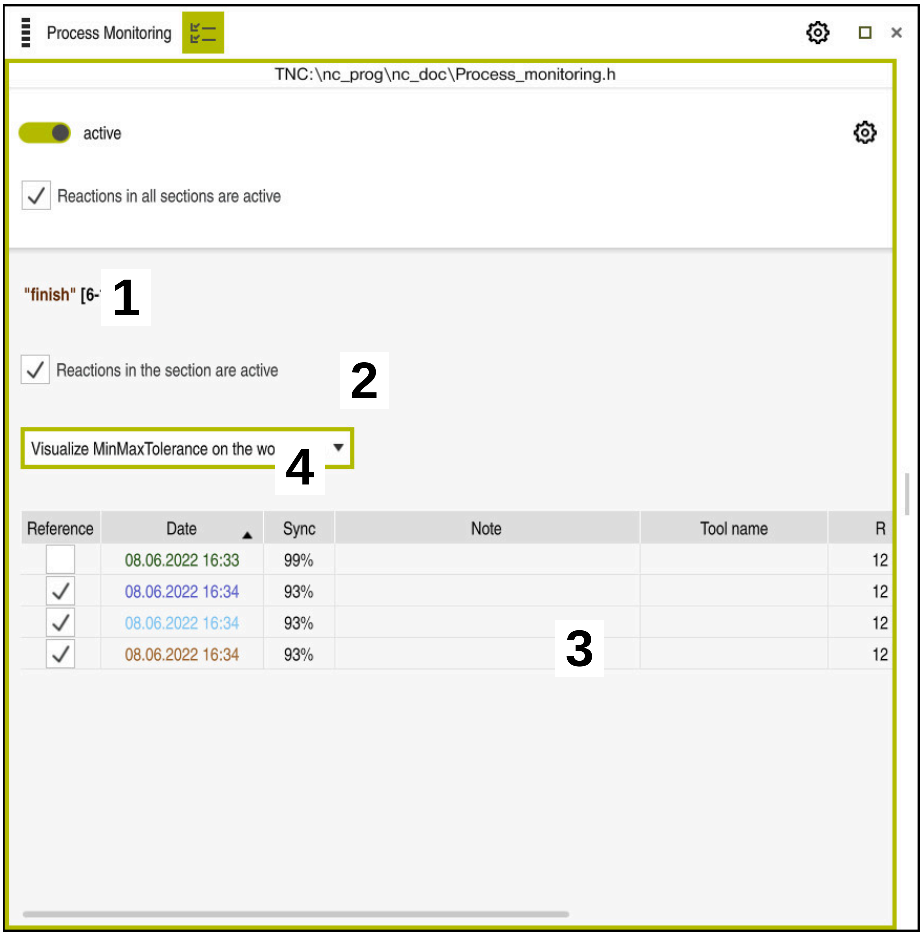
In the global area, the control displays a table with the records of all monitoring sections of the NC program.

Further information: "Records of monitoring sections", Page 1172

Monitoring options column within a monitoring section

If the cursor is inside a monitoring section in the NC program, the **Process Monitoring** workspace displays the **Monitoring options** column within the monitoring section.

If the cursor is within the monitoring section, the control will leave this area gray.



Monitoring options column within the monitoring section

The **Monitoring options** column displays the following within the monitoring section:

1 The control shows the following information and functions:

- Name of the monitoring section, if applicable

  If **AS** is defined in the NC program with the optional syntax element, the control displays the name.

  If no name is defined, the control displays **MONITORING SECTION**.

  Further information: "Input", Page 1175
- Range of NC block numbers of the monitoring section in square brackets

  Start and end of the monitoring section in the NC program

2 Checkbox for activating and deactivating the responses in the monitoring section

You can activate or deactivate the responses of the currently selected monitoring section.

3 Table with the recordings of the monitoring section

The recordings refer only to the monitoring section in which the cursor is currently located.

Further information: "Records of monitoring sections", Page 1172

4 Selection menu for the process heat map

You can display a monitoring task as a process heat map in the **Simulation** workspace.

Further information: "Workpiece options column", Page 1409

Further information: "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1146

Records of monitoring sections

The contents and functions of the table with the recordings of the machining operations depend on the cursor position in the NC program.

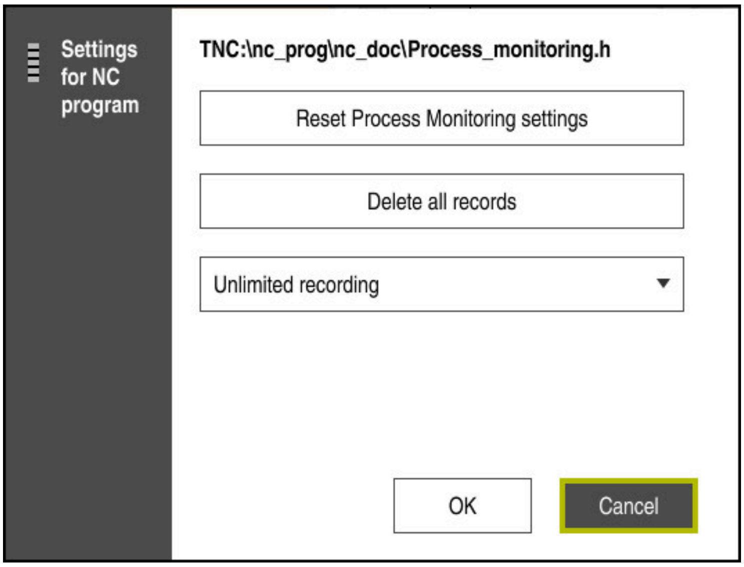
Further information: "Monitoring options column", Page 1169

The table contains the following information about the monitoring section:

Column	Information or action
Reference	If you activate the checkbox for a table row, the control uses this recording as a reference for the corresponding monitoring tasks. If you activate multiple table rows, the control uses all selected rows as references. If you select multiple references with greater deviation, the tunnel width also increases. You can select up to ten references at a time. The effect of the reference depends on the position of the cursor in the NC program: ■ Within the monitoring section: The reference only applies to the currently selected monitoring section. The control shows a dash in the global range in this table row for information. If a table row is marked as a reference in all strategy areas or in the global area, the control displays a check mark. ■ Global area The reference applies to all monitoring sections of the NC program. Mark recordings that have a provided satisfactory result, such as a clean surface, as a reference. You can only select a recording of a complete machining operation as a reference.
Date	Date and time of program start of each recorded machining operation If you select the **Date** column, the control sorts the table by date.
Sync	Quality of synchronization The quality is influenced by the following: ■ Time delay, e.g. change of the feed rate override If the potentiometer position of the feed rate override deviates from the reference machining process, the quality becomes worse. ■ Local delay, e.g. due to a tool correction with **DR** If the path of the tool center point **TCP** deviates from the reference machining operation, the quality becomes worse. The first row of the table is the reference for the quality of the following table rows. **Further information:** "Tool center point (TCP, tool center point)", Page 247 If the quality is between 70% and 80%, the processing operation is still OK. You should manually check the recording in this area.

Column	Information or action
Delete	If you select the trash can icon, the control deletes the table row. You cannot delete the first row of the table because this row is used as a reference for the following functions: ■ **Sync** column ■ **SpindleOverride** monitoring task ■ **FeedOverride** monitoring task You delete all recordings including the first in the **Settings for NC program** window. Only in the global area
Note	In the **Note** column, you can enter notes about the table row.
Tool name	Name of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 270
R	Radius of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 270
DR	Delta value of the tool radius from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 270
L	Length of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 270
CUT	Number of cutting edges of the tool from the tool management Only within the monitoring section **Further information:** "Tool management ", Page 270
CURR_TIME	Tool life from the tool management at the start of the respective machining operation Only within the monitoring section **Further information:** "Tool management ", Page 270

Settings for NC program window



Settings for NC program window

The **Settings for NC program** window provides the following settings:

- **Reset Process Monitoring settings**
- **Delete all records**, including the first table row
- Select the maximum number of recordings of machining operations in the table:
  - **Limit to 5 recordings**
  - **Limit to 10 recordings**
  - **Limit to 50 recordings**
  - **Limit to 200 recordings**
  - **Unlimited recording**

  If the number of machining operations exceeds the maximum number, the control overwrites the last machining operation.

  Further information: "Records of monitoring sections", Page 1172

Notes

- If you use different sizes of workpiece blanks, set process monitoring to a more tolerant setting or start the first monitoring section after pre-machining.
- If the spindle load is too low, the control may not detect any difference from idling, e.g. for a tool with a small diameter.
- If you remove and add a monitoring task again, the previous recordings remain.

Basics of operation

- You can zoom in or out of the graph horizontally by dragging or scrolling.
- If you drag or swipe with the left mouse button held down, you can move the graph.
- You can align the graph by selecting an NC block number. The control marks the selected NC block number in green within the monitoring task.

  Further information: "Common gestures for the touchscreen", Page 96

20.3.3 Defining monitoring sections with MONITORING SECTION (option 168)

Application

With the **MONITORING SECTION** function, you divide the NC program into monitoring sections for process monitoring.

Related topics

- **Process Monitoring** workspace
  Further information: "Process Monitoring workspace (option 168)", Page 1153

Requirement

- Process Monitoring (software option 168)

Description of function

MONITORING SECTION START is used to define the start of a new monitoring section and **MONITORING SECTION STOP**, to define the end of the monitoring section.

You must not nest monitoring sections.

If you do not define a **MONITORING SECTION STOP**, the control still interprets a new monitoring section for the following functions:

- For a new **MONITORING SECTION START**
- For a physical **TOOL CALL**
  The control only interprets a new monitoring section for a tool call when a tool change takes place.
  Further information: "Tool call by TOOL CALL", Page 277

Do not program the following functions within a monitoring section:

- Calling of a subprogram with **LBL CALL**
  Except when the calling subprogram is also programmed within the monitoring section
- Calling of an NC program with **PGM CALL**
- Calling of an NC program with cycle **12 PGM CALL**

You can define monitoring sections within called subprograms or NC programs.

Input

11 MONITORING SECTION START AS "finish contour"	; Start of the monitoring section including additional naming

The NC function includes the following syntax elements:

Syntax element	Meaning
MONITORING SECTION	Syntax initiator for the monitoring section of process monitoring
START or **STOP**	Start or end of the monitoring section
AS	Additional designation Optional syntax element Only when **START** is selected

Notes

- The control shows the start and end of the monitoring section in the structure.
  Further information: "Settings in the Program workspace", Page 197
- End the monitoring section before the end of the program with **MONITORING SECTION STOP**.
  If you do not define an end for the monitoring section, the control ends the monitoring section with **END PGM**.

21

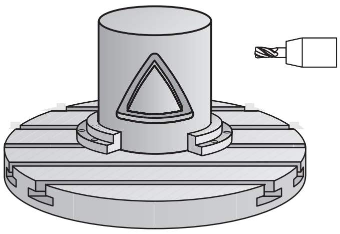
Multiple-Axis Machining

21.1 Cycles for cylinder surface machining

21.1.1 Cycle 27 CYLINDER SURFACE (option 8)

Application

	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to program a contour in two dimensions and then transfer it onto a cylindrical surface. Use Cycle **28** to mill guide slots on the cylinder.

Describe the contour in a subprogram that you program with Cycle **14 CONTOUR**.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

The dimensions in the rotary axis (X coordinates) can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition using **Q17**.

Cycle sequence

1 The control positions the tool above the cutter infeed point, taking the finishing allowance for side into account
2 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate **Q12**.
3 At the end of the contour, the control returns the tool to set-up clearance and returns to the infeed point
4 Steps 1 to 3 are repeated until the programmed milling depth **Q1** is reached.
5 Subsequently, the tool retracts in the tool axis to the clearance height.

	The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Notes

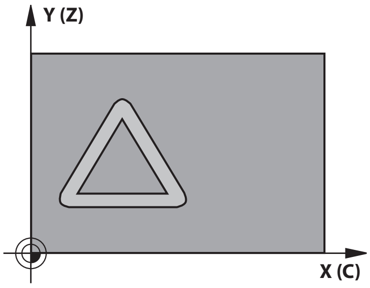
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The memory capacity for programming an SL cycle is limited. You can program up to 16384 contour elements in one SL cycle.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the control will generate an error message. Switching of the kinematics may be required.
- This cycle can also be used in a tilted working plane.

> The machining time can increase if the contour consists of many non-tangential contour elements.

Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	**Q1 Milling depth?** Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**

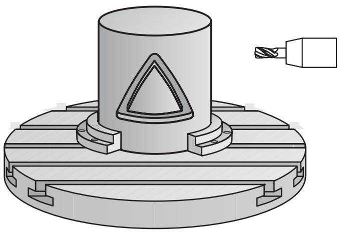
Example

```
11 CYCL DEF 27 CYLINDER SURFACE ~
    Q1=-20              ;MILLING DEPTH ~
    Q3=+0               ;ALLOWANCE FOR SIDE ~
    Q6=+0               ;SET-UP CLEARANCE ~
    Q10=-5              ;PLUNGING DEPTH ~
    Q11=+150            ;FEED RATE FOR PLNGNG ~
    Q12=+500            ;FEED RATE F. ROUGHNG ~
    Q16=+0              ;RADIUS ~
    Q17=+0              ;TYPE OF DIMENSION
```

21.1.2 Cycle 28 CYLINDRICAL SURFACE SLOT (option 8)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



With this cycle you can program a guide slot in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle **27**, with this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the slot are nearly parallel. You can machine exactly parallel walls by using a tool that is exactly as wide as the slot.

The smaller the tool is with respect to the slot width, the larger the distortion in circular arcs and oblique line segments. To minimize this process-related distortion, you can define the parameter **Q21**. This parameter specifies the tolerance with which the control machines a slot as similar as possible to a slot machined with a tool of the same width as the slot.

Program the center path of the contour together with the tool radius compensation. With the radius compensation you specify whether the control cuts the slot with climb milling or up-cut milling.

Cycle sequence

1. The control positions the tool above the infeed point.
2. The control moves the tool vertically to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate **Q12**. The approaching behavior depends on the **ConfigDatum CfgGeoCycle** (no. 201000), **apprDepCylWall** (no. 201004) parameter
3. At the first plunging depth, the tool mills along the programmed slot wall at the milling feed rate **Q12** while respecting the finishing allowance for the side
4. At the end of the contour, the control moves the tool to the opposite slot wall and returns to the infeed point.
5. Steps 2 to 3 are repeated until the programmed milling depth **Q1** is reached.
6. If you defined the tolerance in **Q21**, the control then re-machines the slot walls to be as parallel as possible
7. Finally, the tool retracts in the tool axis to the clearance height.

The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Notes

⚙	This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** machine parameter (no. 201002) to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

NOTICE

Danger of collision!

At the end, the control returns the tool to the set-up clearance, or to the 2nd set-up clearance if one was programmed. The end position of the tool after the cycle need not be the same as the starting position.

- Control the traversing movements of the machine
- In the **Simulation** workspace of the **Editor** operating mode, check the end position of the tool after the cycle.
- After the cycle, program the absolute (not incremental) coordinates

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called.
- This cycle can also be used in a tilted working plane.

ℹ	The machining time can increase if the contour consists of many non-tangential contour elements.

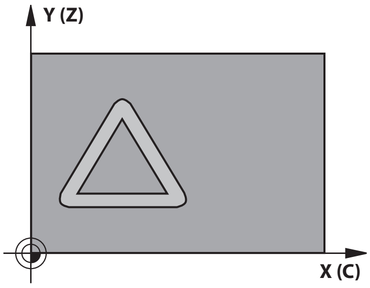
Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Note regarding machine parameters

- Use machine parameter **apprDepCylWall** (no. 201004) to define the approach behavior:
  - **CircleTangential**: Tangential approach and departure
  - **LineNormal**: The tool approaches the contour starting point on a straight line

Cycle parameters

Help graphic	Parameter
	**Q1 Milling depth?** Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance on the slot wall. The finishing allowance reduces the slot width by twice the entered value. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**
	Q20 Slot width? Width of the slot to be machined Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q21 Tolerance? If you use a tool smaller than the programmed slot width **Q20**, process-related distortion occurs on the slot wall wherever the slot follows the path of an arc or oblique line. If you define the tolerance **Q21**, the control adds a subsequent milling operation to ensure that the slot dimensions are as close as possible to those of a slot that has been milled with a tool exactly as wide as the slot. With **Q21**, you define the permitted deviation from this ideal slot. The number of subsequent milling operations depends on the cylinder radius, the tool used, and the slot depth. The smaller the tolerance is defined, the more exact the slot is and the longer the re-machining takes. **Recommendation**: Use a tolerance of 0.02 mm. **Function inactive**: Enter 0 (default setting). Input: **0...9.9999**

Example

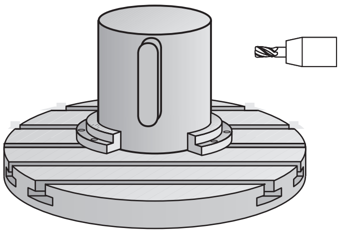
```
11 CYCL DEF 28 CYLINDRICAL SURFACE SLOT ~
    Q1=-20                  ;MILLING DEPTH ~
    Q3=+0                   ;ALLOWANCE FOR SIDE ~
    Q6=+2                   ;SET-UP CLEARANCE ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q12=+500                ;FEED RATE F. ROUGHNG ~
    Q16=+0                  ;RADIUS ~
    Q17=+0                  ;TYPE OF DIMENSION ~
    Q20=+0                  ;SLOT WIDTH ~
    Q21=+0                  ;TOLERANCE
```

21.1.3 Cycle 29 CYL SURFACE RIDGE (option 8)

Application

Refer to your machine manual.

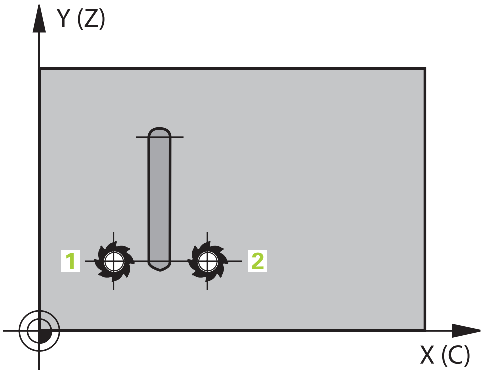
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to program a ridge in two dimensions and then transfer it onto a cylindrical surface. With this cycle, the control adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the center path of the ridge together with the tool radius compensation. With the radius compensation you specify whether the control cuts the ridge with climb milling or up-cut milling.

At the ends of the ridge, the control will always add a semi-circle whose radius corresponds to half the ridge width.

Cycle sequence



1 The control positions the tool above the starting point of machining. The control calculates the starting point from the ridge width and the tool diameter. It is located next to the first point defined in the contour subprogram, offset by half the ridge width and the tool diameter. The radius compensation determines whether machining begins to the left (**1**, RL = climb milling) or to the right of the ridge (**2**, RR = up-cut milling).
2 After the control has positioned the tool to the first plunging depth, the tool moves on a circular arc at the milling feed rate **Q12** tangentially to the ridge wall. A finishing allowance programmed for the side is taken into account.
3 At the first plunging depth, the tool mills along the programmed ridge wall at the milling feed rate **Q12** until the ridge is completed.
4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5 Steps 2 to 4 are repeated until the programmed milling depth **Q1** is reached.
6 Finally, the tool retracts in the tool axis to the clearance height.

> The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Notes

> This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** machine parameter (no. 201002) to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- This cycle requires a center-cut end mill (ISO 1641).
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called. If this is not the case, the control will generate an error message. Switching of the kinematics may be required.

Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Cycle parameters

Help graphic	Parameter
	Q1 Milling depth? Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance on the ridge wall. The finishing allowance increases the ridge width by twice the entered value. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**
	Q20 Ridge width? Width of the ridge to be machined Input: **-99999.9999...+99999.9999**

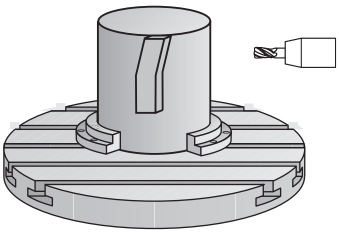
Example

11 CYCL DEF 29 CYL SURFACE RIDGE ~	
Q1=-20	;MILLING DEPTH ~
Q3=+0	;ALLOWANCE FOR SIDE ~
Q6=+2	;SET-UP CLEARANCE ~
Q10=-5	;PLUNGING DEPTH ~
Q11=+150	;FEED RATE FOR PLNGNG ~
Q12=+500	;FEED RATE F. ROUGHNG ~
Q16=+0	;RADIUS ~
Q17=+0	;TYPE OF DIMENSION ~
Q20=+0	;RIDGE WIDTH

21.1.4 Cycle 39 CYL. SURFACE CONTOUR (option 8)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



This cycle enables you to machine a contour on a cylindrical surface. The contour to be machined is programmed on the unrolled surface of the cylinder. With this cycle, the control adjusts the tool in such a way that, with radius compensation active, the walls of the milled contour are always parallel to the cylinder axis.

Describe the contour in a subprogram that you program with Cycle **14 CONTOUR**.

In the subprogram you always describe the contour with the coordinates X and Y, regardless of which rotary axes exist on your machine. This means that the contour description is independent of your machine configuration. The path functions **L**, **CHF**, **CR**, **RND** and **CT** are available.

Unlike in Cycles **28** and **29**, in the contour subprogram, you define the contour actually to be machined.

Cycle sequence

1 The control positions the tool above the starting point of machining. The control locates the starting point next to the first point defined in the contour subprogram offset by the tool diameter
2 The control then moves the tool vertically to the first plunging depth. The tool approaches the workpiece on a tangential path or on a straight line at the milling feed rate **Q12**. A finishing allowance programmed for the side is taken into account. The approach behavior depends on the machine parameter **apprDepCylWall** (no. 201004)
3 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate **Q12** until the contour train is complete.
4 The tool then departs the ridge wall on a tangential path and returns to the starting point of machining.
5 Steps 2 to 4 are repeated until the programmed milling depth **Q1** is reached.
6 Finally, the tool retracts in the tool axis to the clearance height.

> The cylinder must be set up centered on the rotary table. Set the preset to the center of the rotary table.

Notes

> This cycle performs an inclined machining operation. To run this cycle, the first machine axis below the machine table must be a rotary axis. In addition, it must be possible to position the tool perpendicular to the cylinder surface.

NOTICE

Danger of collision!

If the spindle is not switched on when the cycle is called a collision may occur.

- By setting the **displaySpindleErr** machine parameter (no. 201002) to on/off, you can define whether the control displays an error message or not in case the spindle is not switched on.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The spindle axis must be perpendicular to the rotary table axis when the cycle is called.

> - Ensure that the tool has enough space laterally for contour approach and departure.
> - The machining time can increase if the contour consists of many non-tangential contour elements.

Notes on programming

- In the first NC block of the contour program, always program both cylinder surface coordinates.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- The set-up clearance must be greater than the tool radius.
- If you use local **QL** Q parameters in a contour subprogram, you must also assign or calculate these in the contour subprogram.

Note regarding machine parameters

- Use machine parameter **apprDepCylWall** (no. 201004) to define the approach behavior:
  - **CircleTangential**: Tangential approach and departure
  - **LineNormal**: The tool approaches the contour starting point on a straight line

Cycle parameters

Help graphic	Parameter
	Q1 Milling depth? Distance between cylindrical surface and contour floor. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q3 Finishing allowance for side? Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q6 Set-up clearance? Distance between the tool face and the cylindrical surface. This value has an incremental effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q10 Plunging depth? Tool infeed per cut. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q11 Feed rate for plunging? Traversing feed rate in the spindle axis Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q12 Feed rate for roughing? Traversing feed rate in the working plane Input: **0...99999.9999** or **FAUTO**, **FU**, **FZ**
	Q16 Cylinder radius? Radius of the cylinder on which the contour will be machined. Input: **0...99999.9999**
	Q17 Dimension type? deg=0 MM/INCH=1 Program the rotary axis coordinates in degrees or mm (inches) in the subprogram. Input: **0**, **1**

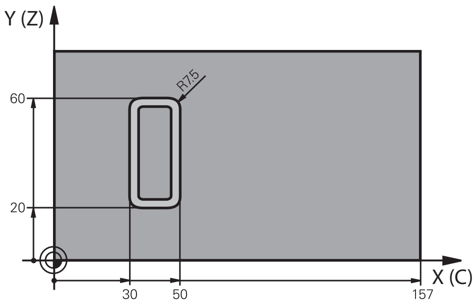
Example

```
11 CYCL DEF 39 CYL. SURFACE CONTOUR ~
    Q1=-20                  ;MILLING DEPTH ~
    Q3=+0                   ;ALLOWANCE FOR SIDE ~
    Q6=+2                   ;SET-UP CLEARANCE ~
    Q10=-5                  ;PLUNGING DEPTH ~
    Q11=+150                ;FEED RATE FOR PLNGNG ~
    Q12=+500                ;FEED RATE F. ROUGHNG ~
    Q16=+0                  ;RADIUS ~
    Q17=+0                  ;TYPE OF DIMENSION
```

21.1.5 Programming Examples

Example: Cylinder surface with Cycle 27

- Machine with B head and C table
- Cylinder centered on rotary table
- Preset is on the underside, in the center of the rotary table

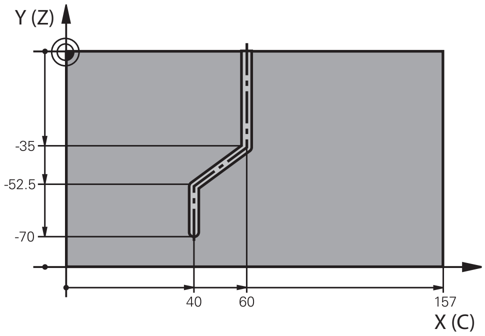


0 BEGIN PGM 5 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call (diameter: 7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 27 CYLINDER SURFACE ~	
Q1=-7 ;MILLING DEPTH ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q10=-4 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+250 ;FEED RATE F. ROUGHNG ~	
Q16=+25 ;RADIUS ~	
Q17=+1 ;TYPE OF DIMENSION	
8 L C+0 R0 FMAX M99	; Pre-position the rotary table, cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 PLANE RESET TURN MB MAX FMAX	; Tilt back, cancel the PLANE function
11 M30	; End of program
12 LBL 1	; Contour subprogram
13 L X+40 Y-20 RL	; Rotary axis data in mm (Q17 = 1)
14 L X+50	
15 RND R7.5	
16 L Y-60	
17 RND R7.5	

18 L IX-20	
19 RND R7.5	
20 L Y-20	
21 RND R7.5	
22 L X+40 Y-20	
23 LBL 0	
24 END PGM 5 MM	

Example: Cylinder surface with Cycle 28

- Cylinder centered on rotary table
- Machine with B head and C table
- Preset is at the center of the rotary table
- Description of the path of the tool center in the contour subprogram



0 BEGIN PGM 4 MM	
1 BLK FORM CYLINDER Z R25 L100	
2 TOOL CALL 3 Z S2000	; Tool call, tool axis (Z), diameter (7)
3 L Z+250 R0 FMAX M3	; Retract the tool
4 PLANE SPATIAL SPA+0 SPB+90 SPC+0 TURN MB MAX FMAX	; Tilt to position
5 CYCL DEF 14.0 CONTOUR	
6 CYCL DEF 14.1 CONTOUR LABEL1	
7 CYCL DEF 28 CYLINDRICAL SURFACE SLOT ~	
Q1=-7 ;MILLING DEPTH ~	
Q3=+0 ;ALLOWANCE FOR SIDE ~	
Q6=+2 ;SET-UP CLEARANCE ~	
Q10=-4 ;PLUNGING DEPTH ~	
Q11=+100 ;FEED RATE FOR PLNGNG ~	
Q12=+250 ;FEED RATE F. ROUGHNG ~	
Q16=+25 ;RADIUS ~	
Q17=+1 ;TYPE OF DIMENSION ~	
Q20=+10 ;SLOT WIDTH ~	
Q21=+0.02 ;TOLERANCE	
8 L C+0 R0 FMAX M99	; Pre-position the rotary table, cycle call
9 L Z+250 R0 FMAX	; Retract the tool
10 PLANE RESET TURN MB MAX FMAX	; Tilt back, cancel the PLANE function
11 M30	; End of program
12 LBL 1	; Contour subprogram, description of the path of the tool center
13 L X+60 Y+0 RL	; Rotary axis data in mm (Q17 = 1)
14 L Y-35	
15 L X+40 Y-52.5	

16 L X-70	
17 LBL 0	
18 END PGM 4 MM	

21.2 Working with the parallel axes U, V and W

21.2.1 Fundamentals

In addition to the principal axes X, Y and Z, the parallel axes U, V and W are available. A parallel axis is, for example, a quill for holes so that smaller masses are moved on large machines.

Further information: "Programmable axes", Page 186

The control provides the following functions for machining with the parallel axes U, V and W:

- **FUNCTION PARAXCOMP**: Define behavior when positioning parallel axes

  Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1195
- **FUNCTION PARAXMODE**: Select three linear axes for machining

  Further information: "Select three linear axes for machining with FUNCTION PARAXMODE", Page 1197

If the machine manufacturer has already enabled the parallel axis in the configuration, the control takes this axis into account in the calculations, without you having to program **PARAXCOMP**. This means that the control continuously takes the parallel axis into account in the calculations and you can therefore also probe a workpiece with any position of the W axis, for example.

In this case, the control displays an icon in the **Positions** workspace.

Further information: "Positions workspace", Page 141

Please note that **PARAXCOMP OFF** does not deactivate the parallel axis in this case, but the control reactivates the standard configuration. The control deactivates automatic calculation only if you include the axis in the NC block, e.g. **PARAXCOMP OFF W**.

After the control has booted, the configuration defined by the machine manufacturer is in effect.

Requirements

- Machine with parallel axes
- Parallel axis functions activated by the machine manufacturer

  The machine manufacturer uses the optional machine parameter **parAxComp** (no. 300205) to define whether the parallel axis function is switched on by default.

21.2.2 Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP

Application

The **FUNCTION PARAXCOMP** function is used to define whether the control takes parallel axes into account in the traversing movements with the associated main axis.

Description of function

If the **FUNCTION PARAXCOMP** function is active, the control displays an icon in the **Positions** workspace. The icon for **FUNCTION PARAXMODE** may cover an active icon for **FUNCTION PARAXCOMP**.

Further information: "Positions workspace", Page 141

FUNCTION PARAXCOMP DISPLAY

Use the **PARAXCOMP DISPLAY** function to activate the display function for parallel axis movements. The control includes movements of the parallel axis in the position display of the associated main axis (sum display). Therefore, the position display of the main axis always displays the relative distance from the tool to the workpiece, regardless of whether you move the main axis or the parallel axis.

FUNCTION PARAXCOMP MOVE

The control uses the **PARAXCOMP MOVE** function to compensate for movements of a parallel axis by performing compensation movements in the associated principal axis.

For example, if a parallel-axis movement is performed in the negative W-axis direction, the principal axis Z is moved simultaneously in the positive direction by the same value. The relative distance from the tool to the workpiece remains the same. Application in gantry-type milling machines: Retract the spindle sleeve to move the cross beam down simultaneously.

FUNCTION PARAXCOMP OFF

Use the **PARAXCOMP OFF** function to switch off the **PARAXCOMP DISPLAY** and **PARAXCOMP MOVE** parallel axis functions.

The following actions cause the control to reset the **PARAXCOMP** parallel-axis function:

- Selection of NC program
- **PARAXCOMP OFF**

When **FUNCTION PARAXCOMP** is not active, the control does not display the corresponding icon and the additional information after the axis designations.

Input

11 FUNCTION PARAXCOMP MOVE W	; Compensate for movements of the W axis by means of a compensating movement in the Z axis

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION PARAXCOMP	Syntax initiator for the behavior when positioning parallel axes
DISPLAY, **MOVE** or **OFF**	Calculate the values of the parallel axis with the main axis, compensate for or do not take into account movements with the main axis
X, **Y**, **Z**, **U**, **V** or **W**	Affected axis Optional syntax element

Note

The **PARAXCOMP MOVE** function can be used only in connection with straight-line blocks (**L**).

21.2.3 Select three linear axes for machining with FUNCTION PARAXMODE

Application

Use the **PARAXMODE** function to define the axes the control is to use for machining. You program all traverse movements and contour descriptions in the principal axes X, Y and Z, independent of your machine.

Requirement

- Parallel axis is calculated

  If your machine manufacturer has not yet activated the **PARAXCOMP** function as default, you must activate **PARAXCOMP** before you can work with **PARAXMODE**.

  Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1195

Description of function

If the **PARAXMODE** function is active, the control uses the axes defined in the function to execute the programmed traverse movements. If the control is to move the principal axis deselected by **PARAXMODE**, you can identify this axis by additionally entering the character **&**. The **&** character then refers to the principal axis.

Further information: "Moving the principal axis and the parallel axis", Page 1198

Define 3 axes in the **PARAXMODE** function (e.g. **FUNCTION PARAXMODE X Y W**) to be used by the control for programmed traverse movements.

If the **FUNCTION PARAXMODE** function is active, the control displays an icon in the **Positions** workspace. The icon for **FUNCTION PARAXMODE** may cover an active icon for **FUNCTION PARAXCOMP**.

Further information: "Positions workspace", Page 141

FUNCTION PARAXMODE OFF

Use the **PARAXCOMP OFF** function to switch off the parallel-axis function. The control then uses the principal axes defined by the machine manufacturer.

The control resets the **PARAXMODE ON** parallel-axis function via the following functions:

- Selection of NC program
- End of program
- **M2** and **M30**
- **PARAXMODE OFF**

Input

11 FUNCTION PARAX MODE X Y W	; Execute programmed traversing movements with axes **X**, **Y** and **W.**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION PARAX MODE	Syntax initiator for axis selection for machining
OFF	Deactivate the parallel axis function Optional syntax element
X, **Y**, **Z**, **U**, **V** or **W**	Three axes for machining Only for **FUNCTION PARAX MODE**

Moving the principal axis and the parallel axis

If the **PARAXMODE** function is active, you can traverse the deselected main axis with the **&** character within the straight line **L**.

Further information: "Straight line L", Page 297

To traverse a deselected main axis:

- Select **L**
- Define coordinates
- Select deselected main axis, e.g. **&Z**
- Enter a value
- Define the radius compensation, if necessary
- Define the feed rate, if necessary
- Define a miscellaneous function, if necessary
- Confirm your input

Notes

- You must deactivate the parallel-axis functions before switching the machine kinematics.
- You can deactivate the programming of parallel axes with the machine parameter **noParaxMode** (no. 105413).
- In order for the control to offset the principal axis deselected with **PARAXMODE**, switch the **PARAXCOMP** function on for this axis.
- Additional positioning of a principal axis with the **&** command is done in the REF system. If you have set the position display to display ACTUAL values, this movement will not be shown. If necessary, switch the position display to REF values.

  Further information: "Position displays", Page 167
- Your machine manufacturer will define the calculation of possible offset values (X_OFFS, Y_OFFS and Z_OFFS from the preset table) for the axes positioned with the **&** operator in the **presetToAlignAxis** machine parameter (no. 300203).

21.2.4 Parallel axes in conjunction with machining cycles

You can also use most machining cycles of the control with parallel axes.

Further information: "Machining Cycles", Page 419

You cannot use the following cycles with parallel axes:

- Cycle **285 DEFINE GEAR** (option 157)
- Cycle **286 GEAR HOBBING** (option 157)
- Cycle **287 GEAR SKIVING** (option 157)
- Touch probe cycles

21.2.5 Example

Drilling is carried out with the W axis in the following NC program:

0 BEGIN PGM PAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S2222	; Call the tool in the tool axis **Z**
4 L Z+100 R0 FMAX M3	; Position the main axis
5 CYCL DEF 200 DRILLING	
Q200=+2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=+150 ;FEED RATE FOR PLNGNG	
Q202=+5 ;PLUNGING DEPTH	
Q210=+0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q211=+0 ;DWELL TIME AT DEPTH	
Q395=+0 ;DEPTH REFERENCE	
6 FUNCTION PARAXCOMP DISPLAY Z	; Activate display compensation
7 FUNCTION PARAXMODE X Y W	; Positive axis selection
8 L X+50 Y+50 R0 FMAX M99	; The parallel axis **W** executes the infeed
9 FUNCTION PARAXMODE OFF	; Restore the standard configuration
10 L M30	
11 END PGM PAR MM	

21.3 Using the facing head with FACING HEAD POS (option 50)

Application

With a facing slide, also called boring head, you can perform almost all turning operations with fewer different tools. The slide position of the facing slide in the X direction can be programmed. On the facing slide you mount, for example, a longitudinal turning tool that you call with a TOOL CALL block.

Related topics

- Machining with parallel axes **U**, **V** and **W**

  Further information: "Working with the parallel axes U, V and W", Page 1195

Requirements

- Combined milling/turning (software option 50)
- Control prepared by the machine manufacturer

  The machine manufacturer must take the facing head into account in the kinematics.
- Kinematics with facing head activated

  Further information: "Switching the operating mode with FUNCTION MODE", Page 210
- Workpiece datum in the working plane is at the center of the rotationally symmetrical contour

  With a facing head, the workpiece datum must not be in the center of the rotary table, because the tool spindle rotates.

  Further information: "Datum shift with TRANS DATUM", Page 972

Description of function

Refer to your machine manual.

The machine manufacturer can provide his own cycles for working with a facing slide. The standard functional range is described below.

The facing head is defined as a turning tool.

Further information: "Turning tool table toolturn.trn (option 50)", Page 1823

Please note for tool calls:

- **TOOL CALL** block without tool axis
- Cutting speed and spindle speed with **TURNDATA SPIN**
- Switch the spindle on with **M3** or **M4**

Machining also works with a tilted working plane and on workpieces that are not rotationally symmetric.

If you move with the facing head without the **FACING HEAD POS** function, you must program the motions of the facing head with the U axis, e.g. in the **Manual operation** application. If the **FACING HEAD POS** function is active, program the facing head with the X axis.

When you activate the facing head, the control automatically positions itself at the workpiece datum in **X** and **Y**. To avoid collisions, you can define a safe height using the **HEIGHT** syntax element.

The facing head is deactivated with the **FUNCTION FACING HEAD** function.

Input

Activating the facing head

11 FACING HEAD POS HEIGHT+100 FMAX	; Activate facing head and move with rapid traverse to safe height **Z**+100

The NC function includes the following syntax elements:

Syntax element	Meaning
FACING HEAD POS	Activate the syntax initiator for the facing head
HEIGHT	Safe height in the tool axis Optional syntax element
F or **FMAX**	Approach safe height with defined feed rate or rapid traverse Optional syntax element
M	Additional function Optional syntax element

Deactivating the facing head

11 FUNCTION FACING HEAD OFF	; Deactivate facing head

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FACING HEAD OFF	Deactivate the syntax initiator for the facing head

Notes

NOTICE

Caution: Danger to the tool and workpiece!

For a facing slide to be used, a kinematic model prepared by the machine manufacturer must be selected by means of the function **FUNCTION MODE TURN**. In this kinematic model, the control implements the programmed X-axis movements of the facing slide as U-axis movements when the **FACING HEAD** function is active. When the **FACING HEAD** function is not active and in **Manual operation** mode, this automated implementation does not take place. As a result, **X** axis movements (programmed or axis key) will be performed in the X axis. In this case, the facing slide must be moved with the U axis. There is a risk of collision during retraction or manual movements!

- Position facing slide at home position with active **FACING HEAD POS** function
- Retract facing slide with active **FACING HEAD POS** function
- In **Manual operation** mode, move the facing slide with the **U** axis key
- Because the **Tilt the working plane** function is possible, pay attention to the 3D ROT status

- To set a spindle speed limitation you can use the **NMAX** value from the tool table as well as **SMAX** value from **FUNCTION TURNDATA SPIN**.
- The following constraints apply to the use of a facing slide:
  - Miscellaneous functions **M91** and **M92** cannot be used
  - Retraction with **M140** is not possible
  - **TCPM** or **M128** are not possible (option 9)
  - **DCM** collision monitoring cannot be used (option 40)
  - Cycles **800**, **801**, and **880** cannot be used
- If you are using the facing slide in the tilted working plane, please note the following:
  - The control calculates the tilted working plane as in milling mode. The **COORD ROT** and **TABLE ROT** functions, as well as **SYM (SEQ)**, refer to the XY plane.

    Further information: "Tilting solution", Page 1016
  - HEIDENHAIN recommends using the **TURN** positioning behavior. The **MOVE** positioning behavior is not the best option in combination with the facing slide.

    Further information: "Rotary axis positioning", Page 1013

21.4 Machining with polar kinematics with FUNCTION POLARKIN

Application

In a polar kinematic model, the path contours of the working plane are performed by one linear axis and one rotary axis instead of by two linear principal axes. The working plane is defined by the linear principal axis and the rotary axis while the working space is defined by these two axes and the infeed axis.

On milling machines, various linear principal axes can be replaced with suitable rotary axes. For example on large machines, polar kinematics enable you to machine much larger surfaces than with only the principal axes.

On turning and grinding machines that have only two linear principal axes, polar kinematics enable milling operations to be performed on the front face.

Requirements

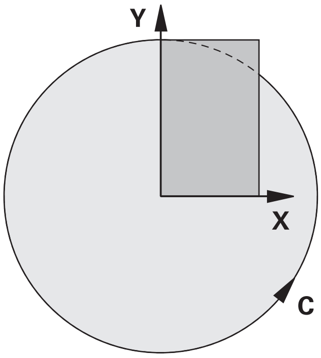
- Machine with at least one rotary axis

  The polar rotary axis must be installed onto the table side so that it is opposite the selected linear axes and must be configured as a modulo axis. Thus, the linear axes must not be positioned between the rotary axis and the table. The maximum range of traverse of the rotary axis is limited by the software limit switches if necessary.
- **PARAXCOMP DISPLAY** function programmed with at least the main axes **X**, **Y** and **Z**.

  HEIDENHAIN recommends defining all of the available axes within the **PARAXCOMP DISPLAY** function.

  Further information: "Defining behavior when positioning parallel axes with FUNCTION PARAXCOMP", Page 1195

Description of function

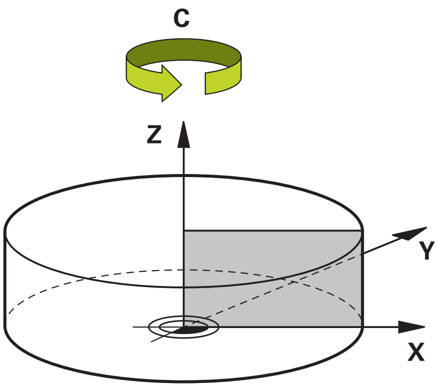


When the polar kinematics are active, the control displays an icon in the **Positions** workspace. This icon covers the icon for the **PARAXCOMP DISPLAY** function.

Use the **POLARKIN AXES** function to activate the polar kinematics. The axis data define the radial axis, the infeed axis, and the polar axis. The **MODE** data influence the positioning behavior, whereas the **POLE** data define the machining at the pole. The pole is the center of rotation of the rotary axis in this case.

Notes on the axes to be selected:

- The first linear axis must be radial to the rotary axis.
- The second linear axis defines the infeed axis and must be parallel to the rotary axis.
- The rotary axis defines the polar axis and is defined last.
- Any available modulo axis that is installed onto the table side so that it is opposite the selected linear axes can be used as the rotary axis.
- The two selected linear axes thus span an area within which the rotary axis also lies.



The following scenarios lead to deactivation of the polar kinematics:

- Execution of the **POLARKIN OFF** function
- Selection of an NC program
- Reaching the end of the NC program
- Abortion of the NC program
- Selecting a kinematic model
- Restarting the control

MODE options

The control provides the following options for positioning behavior:

MODE options:

Syntax	Function
POS	Seen from the center of rotation, the control performs machining in the positive direction of the radial axis. The radial axis must be prepositioned correspondingly.
NEG	Seen from the center of rotation, the control performs machining in the negative direction of the radial axis. The radial axis must be prepositioned correspondingly.
KEEP	The control remains with the radial axis on that side of the center of rotation on which the axis is positioned when the function is activated. If the radial axis is on the center of rotation upon switch-on, **POS** applies.
ANG	The control remains with the radial axis on that side of the center of rotation on which the axis is positioned when the function is activated. If you set **POLE** to **ALLOWED**, positioning through the pole is possible. The pole side is changed and a 180-degree rotation of the rotary axis is prevented.

POLE options

The control provides the following options for machining at the pole:

POLE options:

Syntax	Function
ALLOWED	The control permits machining operations at the pole
SKIPPED	The control prevents machining operations at the pole The disabled area corresponds to a circular surface with a radius of 0.001 mm (1 µm) around the pole.

Input

11 FUNCTION POLARKIN AXES X Z C MODE: KEEP POLE: ALLOWED	; Activate polar kinematics with axes **X**, **Z** and **C**.

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION POLARKIN	Syntax initiator for polar kinematics
AXES or **OFF**	Activate or deactivate polar kinematics
X, **Y**, **Z**, **U**, **V**, **A**, **B**, **C**	Selection of two linear axes and one rotary axis Only when **AXES** is selected Other possibilities might be available, depending on the machine.
MODE:	Selection of the positioning behavior **Further information:** "MODE options", Page 1205 Only when **AXES** is selected
POLE:	Selection of machining in the pole **Further information:** "POLE options", Page 1205 Only when **AXES** is selected

Notes

- The principal axes X, Y, and Z as well as the possible parallel axes U, V, and W can be used as radial axes or infeed axes.
- Position the linear axis that will not be included in the polar kinematics to the coordinate of the pole, before the **POLARKIN** function. Otherwise, a non-machinable area with a radius that corresponds to at least the value of the deselected linear axis would result.
- Avoid performing machining operations at the pole or near the pole, because fluctuations in feed rate may occur in this area. For this reason, ideally use the following **POLE** option: **SKIPPED**.
- The machine manufacturer uses the optional machine parameter **kindOfPref** (no. 202301) to define the behavior of the control when the tool center point path passes through the polar axis.
- Polar kinematics cannot be combined with the following functions:
  - Traverse motions with **M91**

    Further information: "Traversing in the machine coordinate system M-CS with M91", Page 1228
  - Tilting the working plane (option 8)
  - **FUNCTION TCPM** or **M128** (option 9)
- Note that the traversing range of the axes may be limited.

  Further information: "Notes on software limit switches for modulo axes", Page 1218

  Further information: "Traverse limits", Page 1915

21.4.1 Example: SL cycles in the polar kinematics

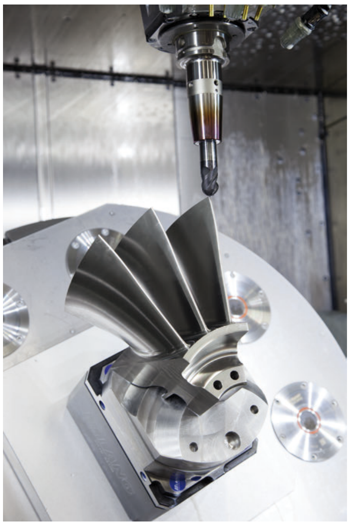
0 BEGIN PGM POLARKIN_SL MM	
1 BLK FORM 0.1 Z X-100 Y-100 Z-30	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 2 Z S2000 F750	
4 FUNCTION PARAXCOMP DISPLAY X Y Z	; Activate **PARAXCOMP DISPLAY**
5 L X+0 Y+0.0011 Z+10 A+0 C+0 FMAX M3	; Pre-position outside the disabled pole area
6 POLARKIN AXES Y Z C MODE:KEEP POLE:SKIPPED	; Activate **POLARKIN**
* - ...	; Datum shift in polar kinematics
9 TRANS DATUM AXIS X+50 Y+50 Z+0	
10 CYCL DEF 7.3 Z+0	
11 CYCL DEF 14.0 CONTOUR	
12 CYCL DEF 14.1 CONTOUR LABEL2	
13 CYCL DEF 20 CONTOUR DATA	
Q1=-10 ;MILLING DEPTH	
Q2=+1 ;TOOL PATH OVERLAP	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q4=+0 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=+2 ;SET-UP CLEARANCE	
Q7=+50 ;CLEARANCE HEIGHT	
Q8=+0 ;ROUNDING RADIUS	
Q9=+1 ;ROTATIONAL DIRECTION	
14 CYCL DEF 22 ROUGH-OUT	
Q10=-5 ;PLUNGING DEPTH	
Q11=+150 ;FEED RATE FOR PLNGNG	
Q12=+500 ;FEED RATE F. ROUGHNG	
Q18=+0 ;COARSE ROUGHING TOOL	
Q19=+0 ;FEED RATE FOR RECIP.	
Q208=+99999 ;RETRACTION FEED RATE	
Q401=+100 ;FEED RATE FACTOR	
Q404=+0 ;FINE ROUGH STRATEGY	
15 M99	
16 CYCL DEF 7.0 DATUM SHIFT	
17 CYCL DEF 7.1 X+0	
18 CYCL DEF 7.2 Y+0	
19 CYCL DEF 7.3 Z+0	
20 POLARKIN OFF	; Deactivate **POLARKIN**
21 FUNCTION PARAXCOMP OFF X Y Z	; Deactivate **PARAXCOMP DISPLAY**
22 L X+0 Y+0 Z+10 A+0 C+0 FMAX	
23 L M30	
24 LBL 2	

25 L X-20 Y-20 RR	
26 L X+0 Y+20	
27 L X+20 Y-20	
28 L X-20 Y-20	
29 LBL 0	
30 END PGM POLARKIN_SL MM	

21.5 CAM-generated NC programs

Application

CAM-generated NC programs are created externally of the control using CAM systems. In combination with 5-axis simultaneous machining and free-form surfaces, CAM systems provide a convenient solution, which in some cases may be the only solution possible.

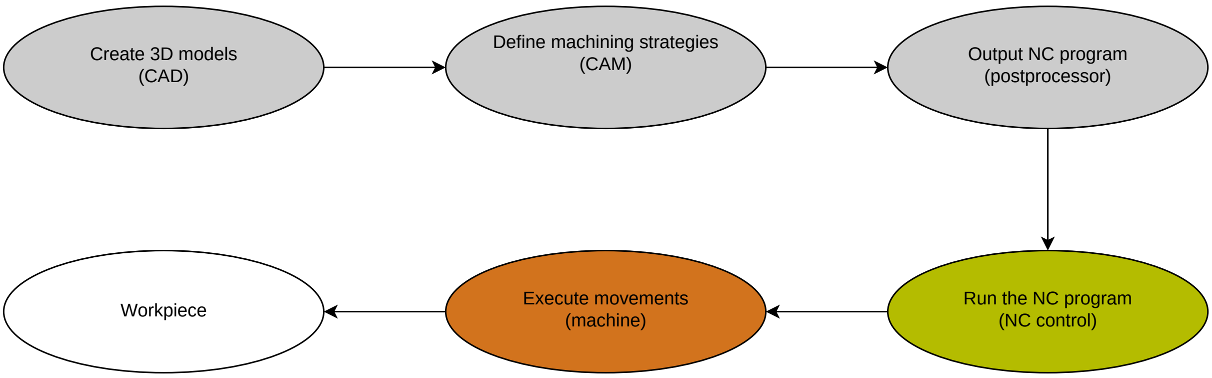


For CAM-generated NC programs to be able to use the full performance potential of the control and to provide you with such options as intervention and correction, certain requirements must be met.

CAM-generated NC programs must meet the same requirements as manually created NC programs. In addition, other requirements arise from the process chain.

Further information: "Process steps", Page 1213

The process chain specifies the path from a design to the finished workpiece.



Related topics

- Using 3D data directly at the control

  Further information: "Opening CAD Files with the CAD-Viewer", Page 1355
- Programming graphically

  Further information: "Graphical Programming", Page 1337

21.5.1 Output formats of NC programs

Output in HEIDENHAIN Klartext format

If you output the NC program in Klartext, you have the following options:

- 3-axis output
- Output with up to five axes, without **M128** or **FUNCTION TCPM**
- Output with up to five axes, with **M128** or **FUNCTION TCPM**

> Requirements for 5-axis machining:
> - Machine with rotary axes
> - Advanced Functions Set 1 (option 8)
> - Advanced Functions Set 2 (option 9) for **M128** or **FUNCTION TCPM**

If the machine kinematics and exact tool data are available to the CAM system, you can output 5-axis NC programs without **M128** or **FUNCTION TCPM**. The programmed feed rate is calculated for all axis components per NC block, which can result in different cutting speeds.

An NC program with **M128** or **FUNCTION TCPM** is machine-neutral and more flexible, since the control takes over the kinematics calculation and uses the tool data from the tool management. The programmed feed rate acts on the tool location point.

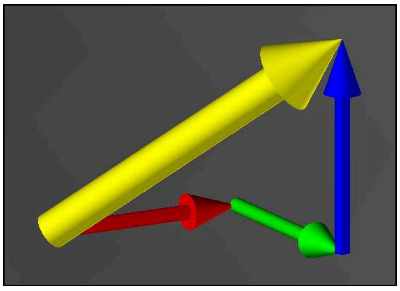
Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027

Further information: "Presets on the tool", Page 244

Examples

11 L X+88 Y+23.5375 Z-8.3 R0 F5000	; 3-axis
11 L X+88 Y+23.5375 Z-8.3 A+1.5 C+45 R0 F5000	; 5-axis without **M128**
11 L X+88 Y+23.5375 Z-8.3 A+1.5 C+45 R0 F5000 M128	; 5-axis with **M128**

Output with vectors



From the point of view of physics and geometry, a vector is a directed variable that describes a direction and a length.

When outputting with vectors, the control requires at least one normalized vector that specifies the direction of the surface normals or the tool position. Optionally, the NC block contains both vectors.

A normalized vector is a vector with the value 1. The vector amount corresponds to the root of the sum of the squares of its components.

$$\sqrt{NX^2 + NY^2 + NZ^2} = 1$$

> Prerequisites:
> - Machine with rotary axes
> - Advanced Functions Set 1 (option 8)
> - Advanced Functions Set 2 (option 9)

> You can only use the output with vectors in milling mode.
> **Further information:** "Switching the operating mode with FUNCTION MODE", Page 210

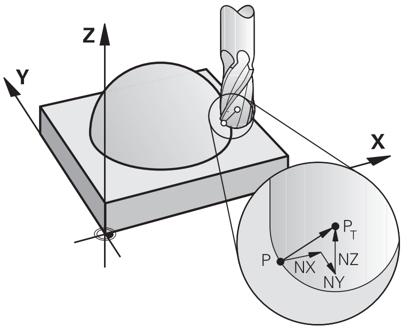
> Vector output with the direction of the surface normals is required for using 3D tool radius compensation depending on the tool's contact angle (option 92).
> **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1062

Examples

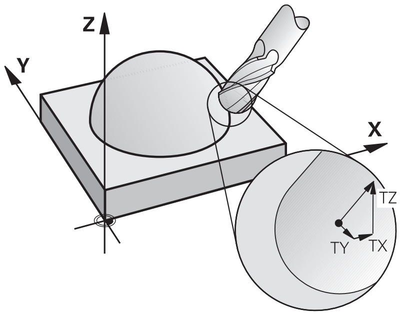
11 LN X0.499 Y-3.112 Z-17.105 NX0.2196165 NY-0.1369522 NZ0.9659258	; 3-axis with surface normal vector, without tool orientation

11 LN X0.499 Y-3.112 Z-17.105 NX0.2196165 NY-0.1369522 NZ0.9659258 TX+0,0078922 TY-0,8764339 TZ+0,2590319 M128	; 5-axis with M128, surface normal vector and tool orientation

Structure of an NC block with vectors



Surface normal vector perpendicular to the contour



Tool direction vector

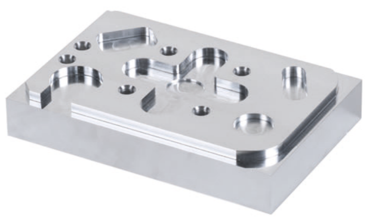
Example

`11 LN X+0.499 Y-3.112 Z-17.105 NX0 NY0 NZ1 TX+0,0078922 TY-0,8764339 TZ+0,2590319`	; Straight line **LN** with surface normal vector and tool orientation

Syntax element	Meaning
LN	Straight line **LN** with surface normal vector
X Y Z	Target coordinates
NX NY NZ	Components of the surface normal vector
TX TY TZ	Components of the tool direction vector

21.5.2 Types of machining according to number of axes

3-axis machining



If only the linear axes **X**, **Y** and **Z** are required for machining a workpiece, 3-axis machining takes place.

3+2-axis machining

If tilting of the working plane is required for machining a workpiece, 3+2-axis machining takes place.

Prerequisites:
- Machine with rotary axes
- Advanced Functions Set 1 (option 8)

Inclined machining

For inclined machining, also referred to as inclined-tool machining, the tool is positioned at a user-defined angle to the working plane. The orientation of the working plane coordinate system **WPL-CS** is not changed, but only the position of the rotary axes and therefore the tool position. The control is able to compensate the offset that is created in the linear axes.

Inclined machining is used in conjunction with undercuts and short tool clamping lengths.

Prerequisites:
- Machine with rotary axes
- Advanced Functions Set 1 (option 8)
- Advanced Functions Set 2 (option 9)

5-axis machining

In 5-axis machining, also referred to as 5-axis simultaneous machining, the machine moves five axes at the same time. For free-form surfaces, this means that the tool can always be oriented perfectly with respect to the workpiece surface.

Prerequisites:
- Machine with rotary axes
- Advanced Functions Set 1 (option 8)
- Advanced Functions Set 2 (option 9)

5-axis machining is not possible with the export version of the control.

21.5.3 Process steps

CAD

Application

Using CAD systems, designers create the 3D models of the required workpieces. Incorrect CAD data has a negative impact on the entire process chain, including the quality of the workpiece.

Notes

- In 3D models, avoid open or overlapping faces and unnecessary points. If possible, use the check functions of the CAD system.
- Design or save the 3D models based on the center of tolerance and not the nominal dimensions.

Support manufacturing with additional files:
- Provide 3D models in STL format. The control-internal simulation can use the CAD data as blank and finished parts, for example. Additional models of tool and workholding equipment are important in conjunction with collision testing (option 40).
- Provide drawings with the dimensions to be checked. The file type of the drawings is not important in this respect, since the control can also open files such as PDFs, and therefore supports paperless production.

Definition

Abbreviation	Definition
CAD (computer-aided design)	Computer-aided design

CAM and postprocessor

Application

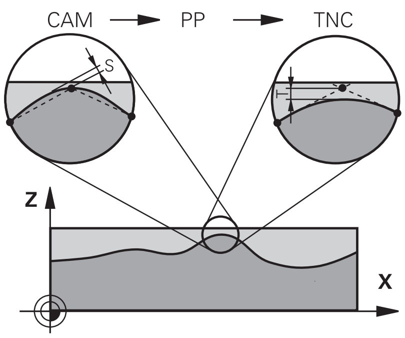
Using machining strategies within the CAM systems, CAM programmers create machine-independent and control-independent NC programs based on the CAD data.

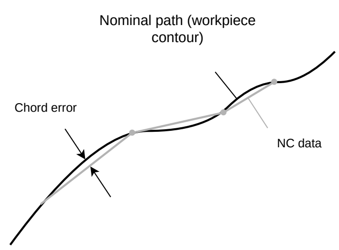
With the aid of the postprocessor, the NC programs are ultimately output specific to machine and control.

Notes on the CAD data

- Avoid quality losses due to unsuitable transfer formats. Integrated CAM systems with manufacturer-specific interfaces work in some cases without loss.
- Take advantage of the available accuracy of the CAD data obtained. A geometry or model error of less than 1 μm is recommended for finishing large radii.

Notes on chord error and Cycle 32 TOLERANCE





- In roughing, the focus is on the processing speed.

  The sum of the chord error and the tolerance **T** in Cycle **32 TOLERANCE** must be smaller than the contour allowance, otherwise contour violations may occur.

Chord error in CAM system	0.004 mm to 0.015 mm
Tolerance **T** in Cycle **32 TOLERANCE**	0.05 mm to 0.3 mm

- When finishing with the aim of high accuracy, the values must provide the required data density.

Chord error in CAM system	0.001 mm to 0.004 mm
Tolerance **T** in Cycle **32 TOLERANCE**	0.002 mm to 0.006 mm

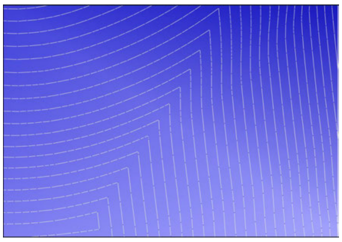
- When finishing with the aim of a high surface quality, the values must allow smoothing of the contour.

Chord error in CAM system	0.001 mm to 0.005 mm
Tolerance **T** in Cycle **32 TOLERANCE**	0.010 mm to 0.020 mm

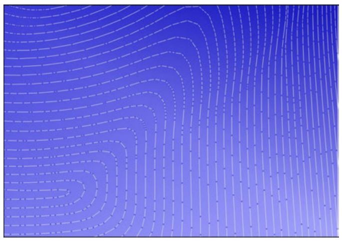
Further information: "Cycle 32 TOLERANCE ", Page 1128

Notes on control-optimized NC output

- Prevent rounding errors by outputting axis positions with at least four decimal places. For optical components and workpieces with large radii (small curves), at least five decimal places are recommended. The output of surface normal vectors (for straight lines **LN**) requires at least seven decimal places.
- You can prevent the cumulation of tolerances by outputting absolute instead of incremental coordinate values for successive positioning blocks.
- If possible, output positioning blocks as arcs. The control calculates circles more accurately internally.
- Avoid repetitions of identical positions, feed specifications and additional functions, e.g. **M3**.
- Output Cycle **32 TOLERANCE** again only when changing settings.
- Make sure that corners (curvature transitions) are precisely defined by an NC block.
- The feed rate fluctuates strongly if the tool path is output with strong changes in direction. If possible, round the tool paths.



Tool paths with strong changes in direction at transitions



Tool paths with rounded transitions

- Do not use intermediate or interpolation points for straight paths. These points are generated, for example, by a constant point output.
- Prevent patterns on the workpiece surface by avoiding exactly synchronous point distribution on surfaces with even curvature.
- Use suitable point distances for the workpiece and the machining step. Possible starting values are between 0.25 mm and 0.5 mm. Values greater than 2.5 mm are not recommended, even with high machining feed rates.
- Prevent mispositioning by outputting the **PLANE** functions (option 8) with **MOVE** or **TURN** without separate positioning blocks. If you output **STAY** and position the rotary axes separately, use variables **Q120** to **Q122** instead of fixed axis values.

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 979
- Prevent strong feed breaks at the tool location point by avoiding an unfavorable relationship between linear and rotary axis motion. A significant change in the tool adjustment angle with a slight change in the position of the tool is a problem, for example. Take into account the different speeds of the axes involved.
- If the machine moves five axes simultaneously, the kinematic errors of the axes may multiply. Use as few axes as possible simultaneously.
- Avoid unnecessary feed rate limits that you can define within **M128** or the **FUNCTION TCPM** (option 9) function for compensation movements.

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027
- Take into account the machine-specific behavior of rotary axes.

  Further information: "Notes on software limit switches for modulo axes", Page 1218

Notes on tools

- A ball-nose cutter, a CAM output to the tool center point and a high rotational axis tolerance **TA** (1° to 3°) in cycle **32 TOLERANCE** enable uniform feed paths.
- Ball-nose or toroidal milling cutter and a CAM output relative to the tool tip require low rotational axis tolerances **TA** (approx. 0.1°) in Cycle **32 TOLERANCE**. Contour violations are more likely to occur at higher values. The extent of the contour violations depends on factors such as the tool position, the tool radius and the depth of engagement.

Further information: "Presets on the tool", Page 244

Notes for user-friendly NC outputs

- Facilitate the easy adaptation of NC programs by using the machining and touch probe cycles of the control.
- Facilitate both the adaptation options and the overview by defining feed rates centrally using variables. It is preferable to use freely usable variables, e.g. **QL** parameters.

  Further information: "Variables: Q, QL, QR and QS parameters", Page 1268
- Provide a better overview by structuring the NC programs. Use e.g. subprograms within the NC programs. If possible, divide larger projects into multiple separate NC programs.

  Further information: "Programming Techniques", Page 331
- Support correction options by outputting contours with tool radius correction.

  Further information: "Tool radius compensation", Page 1038
- Use structure items to enable fast navigation within the NC programs.

  Further information: "Structuring of NC programs", Page 1385
- Use comments to communicate important information about the NC program.

  Further information: "Adding comments", Page 1384

NC control and machine

Application

The control uses the points defined in the NC program to calculate the motions of each machine axis as well as the required velocity profiles. Control-internal filter functions then process and smooth the contour so that the control does not exceed the maximum permissible path deviation.

The motions and velocity profiles calculated are implemented as movements of the tool by the machine's drive system.

You can use various intervention and correction options to optimize machining.

Notes on the use of CAM-generated NC programs

- The simulation of machine and control-independent NC data within the CAM systems can deviate from the actual machining. Check the CAM-generated NC programs using the control-internal simulation.

  Further information: "Simulation Workspace", Page 1405
- Take into account the machine-specific behavior of rotary axes.

  Further information: "Notes on software limit switches for modulo axes", Page 1218
- Make sure that the required tools are available and that the remaining service life is sufficient.

  Further information: "Tool usage test", Page 284
- If necessary, change the values in Cycle **32 TOLERANCE** depending on the chord error and the dynamic response of the machine.

  Further information: "Cycle 32 TOLERANCE ", Page 1128

> Refer to your machine manual.
>
> Some machine manufacturers provide an additional cycle for adapting the behavior of the machine to the respective machining operation (e.g. Cycle **332 Tuning**). Cycle **332** can be used to modify filter settings, acceleration settings and jerk settings.

- If the CAM-generated NC program contains normalized vectors, you can also correct tools three-dimensionally.

  Further information: "Output formats of NC programs", Page 1209

  Further information: "3D radius compensation depending on the tool contact angle (option 92)", Page 1062
- Software options enable further optimizations.

  Further information: "Functions and function packages", Page 1220

  Further information: "Software options", Page 74

Notes on software limit switches for modulo axes

> The following information on software limit switches for modulo axes also applies to traversing limits.
> **Further information:** "Traverse limits", Page 1915

The following general conditions apply to software limit switches for modulo axes:

- The lower limit is greater than –360° and less than +360°.
- The upper limit is not negative and less than +360°.
- The lower limit is not greater than the upper limit.
- The lower and upper limits are less than 360° apart.

If the general conditions are not met, the control cannot move the modulo axis and issues an error message.

If the target position or a position equivalent to it is within the permitted range, movement is permitted with active modulo limit switches. The direction of motion is determined automatically, as only one of the positions can be approached at any one time. Please note the following examples!

Equivalent positions differ by an offset of n x 360° from the target position. The factor n corresponds to any integer.

Example

11 L C+0 R0 F5000	; Limit switches –80° and +80°
12 L C+320	; Target position –40°

The control positions the modulo axis between the active limit switches to the position –40°, which is equivalent to 320°.

Example

11 L C-100 R0 F5000	; Limit switches –90° and +90°
12 L IC+15	; Target position –85°

The control executes the traversing motion because the target position lies within the permitted range. The control positions the axis in the direction of the nearest limit switch.

Example

11 L C-100 R0 F5000	; Limit switches –90° and +90°
12 L IC-15	; Error message

The control issues an error message because the target position is outside the permitted range.

Examples

11 L C+180 R0 F5000	; Limit switches –90° and +90°
12 L C-360	; Target position 0°: Also applies for a multiple of 360°, e.g. 720°

11 L C+180 R0 F5000	; Limit switches –90° and +90°
12 L C+360	; Target position 360°: Also applies for a multiple of 360°, e.g. 720°

If the axis is exactly in the middle of the prohibited area, the distance to both limit switches is identical. In this case, the control can move the axis in both directions.

If the positioning block results in two equivalent target positions in the permitted range, the control positions itself along the shorter path. If both equivalent target positions are 180° away, the control selects the direction of motion according to the programmed algebraic sign.

Definitions

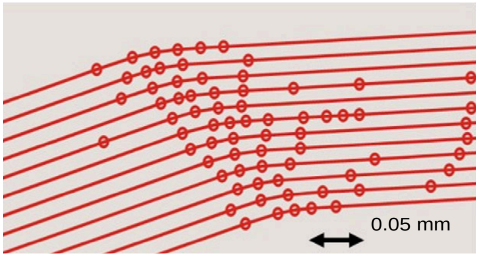
Modulo axis
Modulo axes are axes whose encoder only returns values between 0° and 359.9999°. If an axis is used as a spindle, then the machine manufacturer must configure this axis as a modulo axis.

Rollover axis
Rollover axes are rotary axes that can perform several or any number of revolutions. The machine manufacturer must configure a rollover axis as a modulo axis.

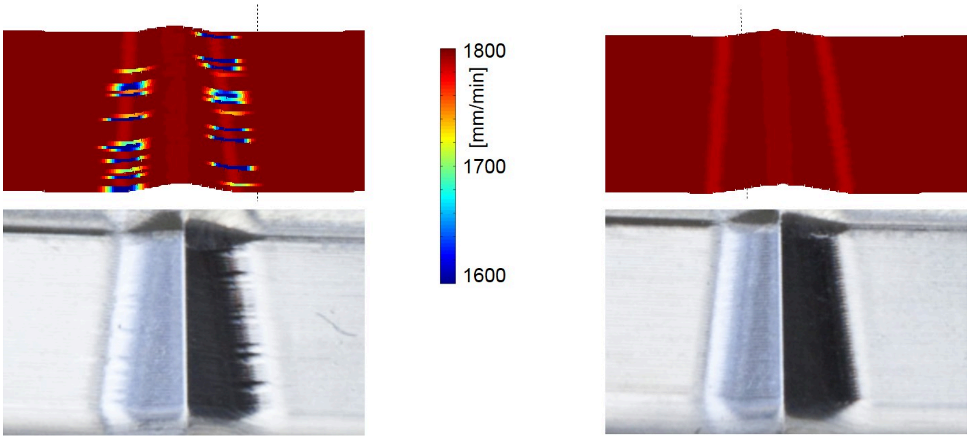
Modulo counting method
The position display of a rotary axis with the modulo counting method is between 0° and 359.9999°. If the value exceeds 359.9999°, the display starts over at 0°.

21.5.4 Functions and function packages

ADP motion control



Distribution of points



Comparison without and with ADP

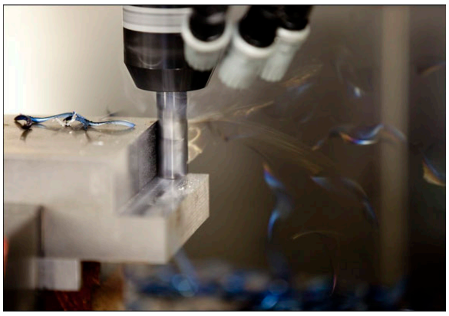
CAM-generated NC programs with an insufficient resolution and variable point density in adjacent paths can lead to feed rate fluctuations and errors on the workpiece surface.

The Advanced Dynamic Prediction (ADP) function extends the prediction of the permissible maximum feed rate profile and optimizes the motion control of the axes involved during milling. This means that you can achieve a high surface quality with a short machining time and reduce the reworking effort.

The most important benefits of ADP at a glance:

- With bidirectional milling, the forward and reverse paths have symmetrical feed behavior.
- Tool paths adjacent to one another have uniform feed paths.
- Negative effects associated with typical problems of CAM-generated NC programs are compensated for or mitigated, e.g.:
  - Short stair-like steps
  - Rough chord tolerances
  - Strong rounded block end point coordinates
- Even under difficult conditions, the control precisely complies with the dynamic parameters.

Dynamic Efficiency



The Dynamic Efficiency package of functions enables you to increase process reliability in heavy machining and roughing in order to improve efficiency.

Dynamic Efficiency includes the following software features:

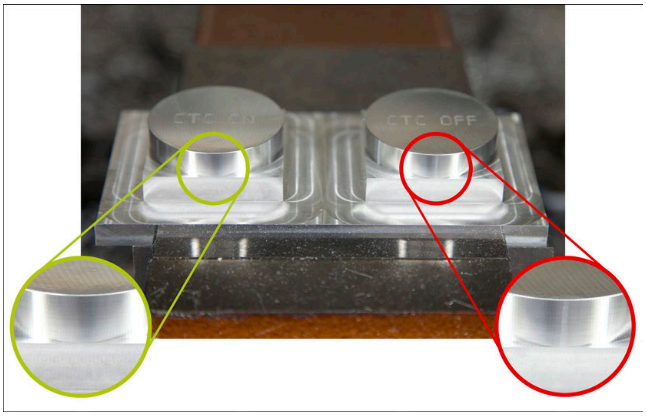
- Active Chatter Control (ACC, option 145)
- Adaptive Feed Control (AFC, option 45)
- Cycles for trochoidal milling (option 167)

Using Dynamic Efficiency offers the following advantages:

- ACC, AFC and trochoidal milling reduce machining time by increasing the material removal rate.
- AFC enables tool monitoring and thus increases process reliability.
- ACC and trochoidal milling extend the tool life.

> You can find more information in the brochure titled **Options and Accessories.**

Dynamic Precision



The Dynamic Precision package of functions enables you to machine quickly and accurately, and with high surface quality.

Dynamic Precision includes the following software functions:

- Cross Talk Compensation (CTC, option 141)
- Position Adaptive Control (PAC, option 142)
- Load Adaptive Control (LAC, option 143)
- Motion Adaptive Control (MAC, option 144)
- Active Vibration Damping (AVD, option 146)

The functions each provide decisive improvements. They can be combined and also mutually complement each other:

- CTC increases the accuracy in the acceleration phases.
- AVD enables better surfaces.
- CTC and AVD result in fast and accurate processing.
- PAC leads to increased contour constancy.
- LAC keeps accuracy constant, even with variable load.
- MAC reduces vibrations and increases the maximum acceleration for rapid traverse movements.

You can find more information in the brochure titled **Options and Accessories.**

22

Miscellaneous Functions

22.1 Miscellaneous functions M and the STOP function

Application

Use miscellaneous functions to activate or deactivate functions of the control and to influence the behavior of the control.

Description of function

You can define up to four miscellaneous functions **M** at the end of an NC block or in a separate NC block. Once you confirm the entry of a miscellaneous function, the control continues with the dialog and you can define additional parameters, such as **M140 MB MAX**.

In the **Manual operation** application, use the **M** button to activate a miscellaneous function.

Further information: "Manual operation application", Page 180

Effects of the miscellaneous functions M

Miscellaneous functions **M** are in effect blockwise or modally. Miscellaneous functions take effect from their point of definition. Other functions or the end of the NC program reset modally effective miscellaneous functions.

Some miscellaneous functions take effect at the start of the NC block and others at the end, regardless of the sequence in which they were programmed.

If you program more than one miscellaneous function in an NC block, the execution sequence is as follows:

- Miscellaneous functions taking effect at the start of the block are executed before those taking effect at the end of the block.
- If more than one miscellaneous function takes effect at the start or end of the block, they are executed in the same sequence as programmed.

STOP function

The **STOP** function interrupts the program run or simulation, e.g., for tool inspection. You can also enter up to four miscellaneous functions **M** in a **STOP** block.

22.1.1 Programming the STOP function

To program the **STOP** function:

STOP

- ▶ Select **STOP**
- > The control creates a new NC block with the **STOP** function.

22.2 Overview of miscellaneous functions

> Refer to your machine manual.
> The machine manufacturer can influence the behavior of the miscellaneous functions described below.
> **M0** to **M30** are standardized miscellaneous functions.

This table shows at what point the miscellaneous functions take effect:

□ At the start of the block

■ At the end of the block

Function	Effect	Further information
M0 Stop program run and the spindle, switch coolant supply off	■	
M1 Optionally stop the program run, optionally stop the spindle, optionally switch the coolant supply off Function depends on the machine manufacturer	■	
M2 Stop program run and the spindle, switch coolant supply off, return to beginning of the program, optionally reset the program information The functions depends on the setting by the machine manufacturer in the machine parameter **resetAt** (no. 100901)	■	
M3 Switch spindle on clockwise	□	
M4 Switch spindle on counterclockwise	□	
M5 Stop the spindle	■	
M6 Tool change; stop program run and spindle	■	

> Since this function depends on the machine manufacturer, HEIDENHAIN recommends using the **TOOL CALL** function for tool changes.
> **Further information:** "Tool call by TOOL CALL", Page 277

Function	Effect	Further information
M8 Switch coolant supply on	□	
M9 Switch coolant supply off	■	
M13 Switch spindle on clockwise, switch coolant supply on	□	
M14 Switch spindle on counterclockwise, switch coolant supply on	□	

Function	Effect	Further information
M30 Function is Identical to **M2**	■	
M89 Free miscellaneous function **or** Call cycle modally Function depends on the machine manufacturer	□ ■	Page 425
M91 Traverse in the machine coordinate system **M-CS**	□	Page 1228
M92 Traverse in the **M92** coordinate system	□	Page 1229
M94 Reduce the display for rotary axes to under 360°	□	Page 1231
M97 Machine small contour steps	■	Page 1233
M98 Machine open contours completely	■	Page 1234
M99 Call a cycle once per block	■	Page 425
M101 Automatically insert a replacement tool	□	Page 1260
M102 Reset **M101**	■	
M103 Reduce feed rate for infeed movements	□	Page 1235
M107 Permit positive tool oversizes	□	Page 1262
M108 Check the radius of the replacement tool Reset **M107**	■	Page 1264
M109 Adapt feed rate for circular paths	□	Page 1236
M110 Reduce feed rate for inner radii	□	
M111 Reset **M109** and **M110**	■	
M116 Interpret feed rate for rotary axes as mm/min	□	Page 1238
M117 Reset **M116**	■	
M118 Activate handwheel superimpositioning	□	Page 1239

Function	Effect	Further information
M120 Pre-calculate the radius-compensated contour (look ahead)	□	Page 1241
M126 Shorter-path traverse of rotary axes	□	Page 1244
M127 Reset **M126**	■	
M128 Automatically compensate for tool inclination (TCPM)	□	Page 1245
M129 Reset **M128**	■	
M130 Traverse in the non-tilted input coordinate system **I-CS**	□	Page 1230
M136 Interpret feed rate as mm/rev	□	Page 1250
M137 Reset **M136**	■	
M138 Take rotary axes into account during machining operations	□	Page 1251
M140 Retract in the tool axis	□	Page 1252
M141 Suppress touch probe monitoring	□	Page 1264
M143 Rescind basic rotations	□	Page 1254
M144 Factor the tool offset into the calculations	□	Page 1255
M145 Reset **M144**	■	
M148 Automatically lift off upon an NC stop or a power failure	□	Page 1257
M149 Reset **M148**	■	
M197 Prevent rounding off of outside corners	■	Page 1258

22.3 Miscellaneous functions for coordinate entries

22.3.1 Traversing in the machine coordinate system M-CS with M91

Application

You can use **M91** to program machine-based positions, such as for moving to safe positions. The coordinates of positioning blocks with **M91** are effective in the machine coordinate system **M-CS**.

Further information: "Machine coordinate system M-CS", Page 935

Description of function

Effect

M91 is in effect blockwise and takes effect at the start of the block.

Application example

11 LBL "SAFE"	
12 L Z+250 R0 FMAX M91	; Approach a safe position in the tool axis
13 L X-200 Y+200 R0 FMAX M91	; Approach a safe position in the plane
14 LBL 0	

Here **M91** is in a subprogram in which the control moves the tool to a safe position, by first moving in the tool axis and then in the plane.

Since the coordinates refer to the machine datum, the tool always moves to the same position. That way, regardless of the workpiece preset, the subprogram can be repeatedly called in the NC program, for example before tilting the rotary axes.

Without **M91** the control references the programmed coordinates to the workpiece preset.

Further information: "Presets in the machine", Page 187

The coordinates for a safe position depend on the machine.
The machine manufacturer defines the position of the machine datum.

Notes

- If you program incremental coordinates in an NC block with the miscellaneous function **M91**, then these coordinates are relative to the last position programmed with **M91**. For the first position programmed with **M91**, the incremental coordinates are relative to the current tool position.
- The control considers any active tool radius compensation when positioning with **M91**.

  Further information: "Tool radius compensation", Page 1038
- The control uses the tool carrier reference point when positioning in the tool axis.

  Further information: "Presets in the machine", Page 187
- The following position displays refer to the machine coordinate system **M-CS** and show the values defined with **M91**:
  - **Nominal reference position (RFNOML)**
  - **Actual reference position (RFACTL)**

  Further information: "Position displays", Page 167
- In the **Editor** operating mode, use the **Workpiece position** window to apply the current workpiece preset for the simulation. In this constellation you can simulate traverse movements with **M91**.

  Further information: "Visualization options column", Page 1407
- In the machine parameter **refPosition** (no. 400403) the machine manufacturer defines the position of the machine datum.

22.3.2 Traversing in the M92 coordinate system with M92

Application

You can use **M92** to program machine-based positions, such as for moving to safe positions. The coordinates of positioning blocks with **M92** are relative to the **M92** datum and are effective in the **M92** coordinate system.

Further information: "Presets in the machine", Page 187

Description of function

Effect

M92 is in effect blockwise and takes effect at the start of the block.

Application example

11 LBL "SAFE"	
12 L Z+0 R0 FMAX M92	; Approach a safe position in the tool axis
13 L X+0 Y+0 R0 FMAX M92	; Approach a safe position in the plane
14 LBL 0	

Here **M92** is in a subprogram in which the tool moves to a safe position, by first moving in the tool axis and then in the plane.

Since the coordinates refer to the **M92** datum, the tool always moves to the same position. That way, regardless of the workpiece preset, the subprogram can be repeatedly called in the NC program, for example before tilting the rotary axes.

Without **M92** the control references the programmed coordinates to the workpiece preset.

Further information: "Presets in the machine", Page 187

> The coordinates for a safe position depend on the machine.
> The machine manufacturer defines the position of the **M92** datum.

Notes

- The control considers any active tool radius compensation when positioning with **M92**.

  Further information: "Tool radius compensation", Page 1038
- The control uses the tool carrier reference point when positioning in the tool axis.

  Further information: "Presets in the machine", Page 187
- In the **Editor** operating mode, use the **Workpiece position** window to apply the current workpiece preset for the simulation. In this constellation you can simulate traverse movements with **M92**.

  Further information: "Visualization options column", Page 1407
- In the optional machine parameter **distFromMachDatum** (no. 300501) the machine manufacturer defines the position of the **M92** datum.

22.3.3 Traversing in the non-tilted input coordinate system I-CS with M130

Application

Coordinates of a straight line entered with **M130** are effective in the non-tilted input coordinate system **I-CS** despite a tilted working plane, such as for retraction.

Description of function

Effect

M130 is in effect blockwise for straight lines without radius compensation and takes effect at the start of the block.

Further information: "Straight line L", Page 297

Application example

11 L Z+20 R0 FMAX M130	; Retract in the tool axis

With **M130**, the control references the coordinates in this NC block to the non-tilted input coordinate system **I-CS** despite a tilted working plane. That way the control retracts the tool perpendicular to the top edge of the workpiece.

Without **M130** the control references the coordinates of the straight line to the tilted **I-CS**.

Further information: "Input coordinate system I-CS", Page 944

Notes

NOTICE

Danger of collision!

The miscellaneous function **M130** is effective only blockwise. The control executes the subsequent machining operations in the tilted working plane coordinate system **WPL-CS** again. Danger of collision during machining!

- Use the simulation to check the sequence and positions

If you combine **M130** with a cycle call, the control will interrupt machining with an error message.

Definition

Non-tilted input coordinate system I-CS
In a non-tilted input coordinate system **I-CS** the control ignores the tilting of the working plane, but does take into account the alignment of the workpiece's upper surface and all active transformations, such as a rotation.

22.4 Miscellaneous functions for path behavior

22.4.1 Reducing the display for rotary axes to under 360° with M94

Application

With **M94** the control reduces the display of the rotary axes to a range between 0° and 360°. Additionally, this limitation reduces the angle difference between the actual position and the new nominal position to less than 360°, which shortens traverse movements.

Related topics

- Values of the rotary axes in the position display

  Further information: "Positions workspace", Page 141

Description of function

Effect

M94 is in effect blockwise and takes effect at the start of the block.

Application example

11 L IC+420	; Move the C axis
12 L C+180 M94	; Reduce the display value of the C axis and move the axis

Before machining, the control shows the value 0° in the position display of the C axis.

In the first NC block the C axis moves incrementally by 420°, for example in order to cut an adhesive slot.

The second NC block first reduces the display of the C axis from 420° to 60°. Then the control positions the C axis to the nominal position of 180°. The angle difference is now 120°.

Without **M94** the angle difference would be 240°.

Input

If you define **M94**, the control continues the dialog and prompts you for the affected rotary axis. If you do not enter an axis, the control reduces the position display for all rotary axes.

21 L M94	; Reduce the display values of all rotary axes

21 L M94 C	; Reduce the display value of the C axis

Notes

- **M94** only affects rollover axes whose actual position display permits values above 360°.
- In the machine parameter **isModulo** (no. 300102) the machine manufacturer defines whether the modulo counting method is used for a rollover axis.
- In the optional machine parameter **shortestDistance** (no. 300401) the machine manufacturer defines whether the control by default positions the rotary axis on the shortest traverse path.
- In the optional machine parameter **startPosToModulo** (no. 300402) the machine manufacturer defines whether the control reduces the actual position display to a range between 0° and 360° before each positioning.
- If traverse limits or software limit switches are active for a rotary axis then **M94** has no effect on this rotary axis.

Definitions

Modulo axis

Modulo axes are axes whose encoder only returns values between 0° and 359.9999°. If an axis is used as a spindle, then the machine manufacturer must configure this axis as a modulo axis.

Rollover axis

Rollover axes are rotary axes that can perform several or any number of revolutions. The machine manufacturer must configure a rollover axis as a modulo axis.

Modulo counting method

The position display of a rotary axis with the modulo counting method is between 0° and 359.9999°. If the value exceeds 359.9999°, the display starts over at 0°.

22.4.2 Machining small contour steps with M97

Application

With **M97** you can produce contour steps that are smaller than the tool radius. The control does not damage the contour and does not issue an error message.

> Instead of **M97**, HEIDENHAIN recommends using the more powerful function **M120** (option 21).
>
> After activating **M120** you can produce complete contours without error messages. **M120** also considers circular paths.

Related topics

- Pre-calculating a radius-compensated contour with **M120**

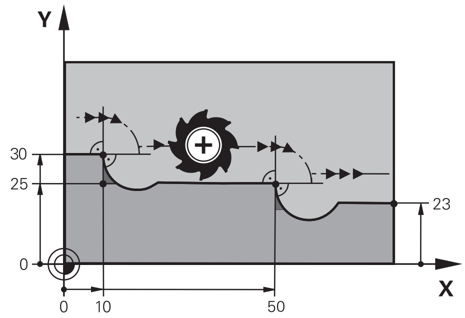
  Further information: "Pre-calculating a radius-compensated contour with M120", Page 1241

Description of function

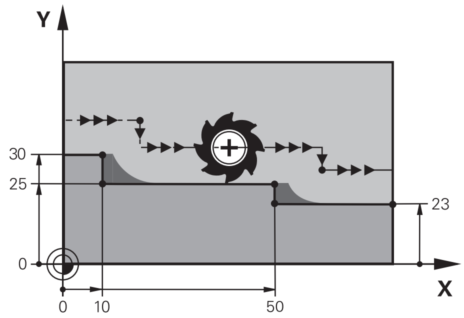
Effect

M97 is in effect blockwise and takes effect at the end of the block.

Application example



Contour step without **M97**



Contour step with **M97**

11 TOOL CALL 8 Z S5000	; Insert the tool with diameter 16
*** - ...**	
21 L X+0 Y+30 RL	
22 L X+10 M97	; Machine the contour step using the path intersection
23 L Y+25	
24 L X+50 M97	; Machine the contour step using the path intersection
25 L Y+23	
26 L X+100	

For radius-compensated contour steps, the control uses **M97** to determine a path intersection that is in the extension of the tool path. The control extends the tool path each time by the tool radius. This means that the smaller the counter step is and the larger the tool radius, the greater the contour extension is. The control moves the tool beyond the path intersection and thus avoids damage to the contour.

Without **M97** the tool would move on a transitional arc around the outside corners and damage the contour. At such locations the control interrupts machining with the **Tool radius too large** error message.

Notes

- Program **M97** only for outside corners.
- For further machining operations, please note that shifting the contour corner results in more residual material. You may then need to rework the contour step with a smaller tool.

22.4.3 Machining open contour corners with M98

Application

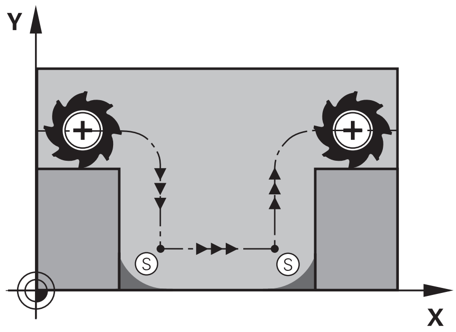
If the tool performs a machining operation on a radius-compensated contour, then residual material remains at the inside corners. With **M98** the control extends the tool path by the tool radius so that the tool completely machines an open contour and removes all residual material.

Description of function

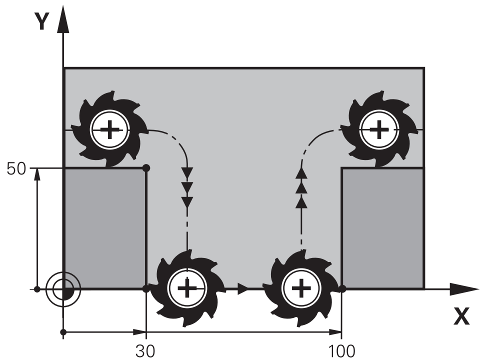
Effect

M98 is in effect blockwise and takes effect at the end of the block.

Application example



Open contour without **M98**



Open contour with **M98**

11 L X+0 Y+50 RL F1000	
12 L X+30	
13 L Y+0 M98	; Completely machine an open contour corner
14 L X+100	; The control maintains the position of the Y axis with **M98**
15 L Y+50	

The control moves the tool along the contour with radius compensation. With **M98** the control calculates the contour ahead of time and determines a new path intersection in the extension of tool path. The control moves the tool beyond this path intersection and completely machines the open contour.

In the next NC block the control maintains the position of the Y axis.

Without **M98** the control uses the programmed coordinates as limitation for the radius-compensated contour. The control calculates the path intersection so that the contour is not damaged and residual material remains.

22.4.4 Reducing the feed rate for infeed movements with M103

Application

With **M103** the control performs infeed movements at a lower feed rate, for example when plunging. You use a percent factor to define the feed-rate value.

Description of function

Effect

M103 is in effect for straight lines in the tool axis at the start of the block.

In order to reset **M103**, program **M103** without a defined factor.

Application example

11 L X+20 Y+20 F1000	; Move in the working plane
12 L Z-2.5 M103 F20	; Activate feed rate reduction and move at reduced feed rate
12 L X+30 Z-5	; Move at reduced feed rate

In the first NC block the control positions the tool in the working plane.

In NC block **12** the control activates **M103** with the percent factor 20 and then performs the infeed movement in the Z axis at a reduced feed rate of 200 mm/min.

Next, in NC block **13**, the control performs an infeed movement in the X and Z axes at a reduced feed rate of 825 mm/min. This higher feed rate results from the control moving the tool in the plane in addition to the infeed movement. The control calculates a cutting value between the feed rate in the plane and the infeed rate.

Without **M103** the infeed movement is performed at the programmed feed rate.

Input

If you define **M103**, the control continues the dialog and prompts you for the factor **F**.

Notes

- The infeed rate F_Z is calculated from the last programmed feed rate F_{Prog} and the percent factor **F**.

  $F_Z = F_{Prog} \times F$
- **M103** is also effective with an active tilted working plane coordinate system **WPL-CS**. The feed rate reduction is then effective during infeed movements in the virtual tool axis **VT**.

22.4.5 Adapting the feed rate for circular paths with M109

Application

With **M109** the control maintains a constant feed rate at the cutting edge for internal and external machining on circular paths, for example to produce a uniform milled surface during finishing.

Description of function

Effect

M109 takes effect at the start of the block.

In order to reset **M109**, program **M111**.

Application example

11 L X+5 Y+25 RL F1000	; Approach first contour point at programmed feed rate
12 CR X+45 Y+25 R+20 DR- M109	; Activate feed rate adaptation, then perform the operation on the circular path at the increased feed rate

In the first NC block the control moves the tool at the programmed feed rate, which refers to the tool center-point path.

In NC block **12** the control activates **M109** and maintains a constant feed rate at the tool cutting edge when machining on circular paths. At the beginning of each block the control calculates the feed rate at the tool cutting edge for the respective NC block and adapts the programmed feed rate depending on the contour radius and tool radius. This means that the programmed feed rate is increased for external operations and reduced for internal operations.

The tool then cuts the external contour at an increased feed rate.

Without **M109** the tool cuts along the circular path at the programmed feed rate.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

If the **M109** function is active, the control might significantly increase the feed rate when machining very small outside corners (acute angles). There is a risk of tool breakage or workpiece damage during machining.

- Do not use **M109** for machining very small outside corners (acute angles)

If you define **M109** before calling a machining cycle with a number greater than **200**, the adjusted feed rate is also effective for circular paths within these machining cycles.

22.4.6 Reducing the feed rate for internal radii with M110

Application

With **M110** the control maintains a constant feed rate at the cutting edge only for internal radii, as opposed to **M109**. This results in consistent cutting conditions affecting the tool, which is important, for example, in heavy-duty machining.

Description of function

Effect

M110 takes effect at the start of the block.

In order to reset **M110**, program **M111**.

Application example

11 L X+5 Y+25 RL F1000	; Approach first contour point at programmed feed rate
12 CR X+45 Y+25 R+20 DR+ M110	; Activate feed rate reduction, then perform the operation on the circular path at the reduced feed rate

In the first NC block the control moves the tool at the programmed feed rate, which refers to the tool center-point path.

In NC block **12** the control activates **M110** and maintains a constant feed rate at the tool cutting edge when machining on internal radii. At the beginning of each block the control calculates the feed rate at the tool cutting edge for the respective NC block and adapts the programmed feed rate depending on the contour radius and tool radius.

The tool then cuts the internal radius at a reduced feed rate.

Without **M110** the tool cuts along the internal radius at the programmed feed rate.

Note

If you define **M110** before calling a machining cycle with a number greater than **200**, the adjusted feed rate is also effective for circular paths within these machining cycles.

22.4.7 Interpreting the feed rate for rotary axes as mm/min with M116 (option 8)

Application

With **M116** the control interprets the feed rate for rotary axes as millimeters per minute.

Requirements

- Machine with rotary axes
- Kinematics description

> Refer to your machine manual.
> The machine manufacturer creates the kinematics description of the machine.

- Software option 8: Advanced Functions (set 1)

Description of function

Effect

M116 is effective only in the working plane and takes effect at the start of the block.

In order to reset **M116**, program **M117**.

Application example

11 L IC+30 F500 M116	; Move in the C axis in mm/min

With **M116** the control interprets the programmed feed rate of the C axis as mm/min, such as for cylinder surface machining.

In this case, the control calculates the feed for the block at the start of each NC block, taking the distance from the tool center point to the center of the rotary axis into account.

The feed rate does not change while the control is executing the NC block. This also applies for when the tool is moving towards the center of a rotary axis.

Without **M116** the control interprets the feed rate programmed for a rotary axis as degrees per minute.

Notes

- You can program **M116** for head and table rotary axes.
- The **M116** function also has an effect if the **Tilt working plane** function is active.

  Further information: "Tilting the working plane (option 8)", Page 978
- It is not possible to combine **M116** with **M128** or **FUNCTION TCPM** (option 9). If you want to activate **M116** for an axis while **M128** or **FUNCTION TCPM** is active, then you must use **M138** to exclude this axis before machining.

  Further information: "Taking rotary axes into account during machining operations with M138", Page 1251
- Without **M128** or **FUNCTION TCPM** (option 9), **M116** can take effect for multiple rotary axes at the same time.

22.4.8 Activating handwheel superimpositioning with M118

Application

With **M118** the control activates handwheel superimpositioning. You can then perform manual corrections by handwheel during program run.

Related topics

- Handwheel superimpositioning with global program settings GPS (option 44)

Requirements

- Handwheel
- Software option 21: Advanced Functions (set 3)

Description of function

Effect

M118 takes effect at the start of the block.

In order to reset **M118**, program **M118** without entering any axes.

> ℹ Canceling a program also resets handwheel superimpositioning.

Application example

11 L Z+0 R0 F500	; Move in the tool axis
12 L X+200 R0 F250 M118 Z1	; Move in the working plane with active handwheel superimpositioning of no more than ±1 mm in the Z axis

In the first NC block the control positions the tool in the tool axis.

In NC block **12** the control activates handwheel superimpositioning at the start of the block with a maximum traverse range of ±1 mm in the Z axis.

Then the control performs the traverse movement in the working plane. During this traverse movement you can use the handwheel for continuous motion of the tool in the Z axis by up to ±1 mm. This way you can, for example, rework a workpiece that has been reclamped but that cannot be probed due to its free-form surface.

Input

If you define **M118**, the control continues the dialog and prompts you for the axes and the maximum permissible superimpositioning value. For linear axes you define the value in millimeters and for rotary axes in degrees.

21 L X+0 Y+38.5 RL F125 M118 X1 Y1	; Move in the working plane with active handwheel superimpositioning of no more than ±1 mm in the X and Y axes

Notes

> Refer to your machine manual.
>
> Your machine manufacturer must have prepared the control for this function.

- By default **M118** is effective in the machine coordinate system **M-CS**.

  If you activate the **handwheel superimpositioning** switch in the **GPS** (option 44) workspace, handwheel superimpositioning is effective in the last selected coordinate system.

  Further information: "Global Program Settings (GPS, option 44)", Page 1132
- On the **POS HR** tab of the **Status** workspace the control shows the active coordinate system in which handwheel superimpositioning is in effect, as well as the maximum possible traverse values of the respective axes.

  Further information: "POS HR tab", Page 158
- The handwheel superimpositioning function with **M118** in combination with dynamic collision monitoring (DCM, option 40) is possible only at a standstill.

  To be able to use **M118** without restrictions, you have to deactivate **DCM** (option 40) or activate a kinematics model without collision objects.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084
- Handwheel superimpositioning is also effective in the **MDI** application.

  Further information: "Application MDI", Page 1759
- If you want to use **M118** with clamped axes, you must unclamp them first.

Notes in conjunction with the virtual tool axis VT (option 44)

Refer to your machine manual.
Your machine manufacturer must have prepared the control for this function.

- On machines with head rotation axes, you can choose for inclined machining whether superimpositioning should be in effect in the Z axis or along the virtual tool axis **VT**.
- In the machine parameter **selectAxes** (no. 126203) the machine manufacturer defines the assignment of axis keys on the handwheel.

  When using an HR 5xx handwheel, you can assign the virtual axis to the orange **VI** axis key, if desired.

22.4.9 Pre-calculating a radius-compensated contour with M120

Application

With **M120** the control pre-calculates a radius-compensated contour. This way the control can produce contours that are smaller than the tool radius without damaging the contour or issuing an error message.

Requirement

- Software option 21: Advanced Functions (set 3)

Description of function

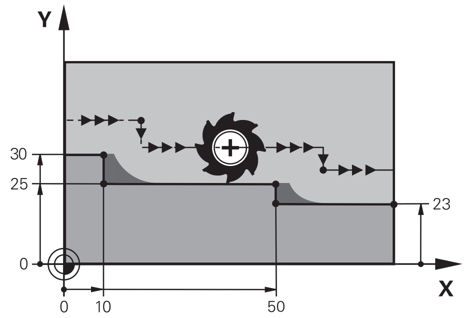
Effect

M120 takes effect at the start of the block and remains in effect beyond the milling cycles.

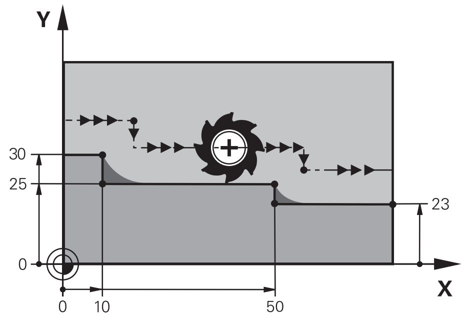
You can reset **M120** with the following functions:

- Radius compensation **R0**
- **M120 LA0**
- **M120** without **LA**
- **PGM CALL**
- Cycle **19 WORKING PLANE** or **PLANE** functions (option 8)

Application example



Contour step with **M97**



Contour step with **M120**

11 TOOL CALL 8 Z S5000	; Insert the tool with diameter 16
*** - ...**	
21 L X+0 Y+30 RL M120 LA2	; Activate contour pre-calculation and move in the working plane
22 L X+10	
23 L Y+25	
24 L X+50	
25 L Y+23	
26 L X+100	

With **M120 LA2** in NC block **21**, the control checks the radius-compensated contour for undercuts. In this example the control calculates the tool path starting from the current NC block for two NC blocks at a time. Then the control uses radius compensation while positioning the tool to the first contour point.

When machining the contour, the control extends the tool path in each case so that the tool does not damage the contour.

Without **M120** the tool would move on a transitional arc around the outside corners and damage the contour. At such locations the control interrupts machining with the **Tool radius too large** error message.

Input

If you define **M120**, the control continues the dialog and prompts you for the number of **LA** NC blocks to be calculated in advance (up to 99).

Notes

NOTICE

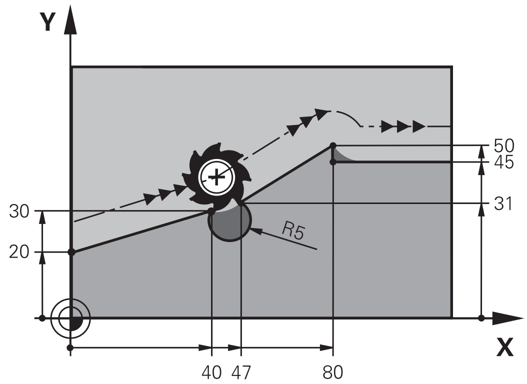
Danger of collision!

Define as low a number as possible of **LA** NC blocks to be pre-calculated. If the value defined is too large, the control might overlook parts of the contour!

- Use the Simulation mode to test the NC program before execution
- Verify the NC program by slowly executing it block by block

- For further machining operations, please note that residual material remains in the contour corners. You may then need to rework the contour step with a smaller tool.
- If you always program **M120** in the same NC block as the radius compensation you can achieve consistent and clearly structured programs.
- If you run the following functions while **M120** is active, then the control cancels program run and issues an error message:
  - Cycle **32 TOLERANCE**
  - **M128** (option 9)
  - **FUNCTION TCPM** (option 9)
  - Mid-program startup

Example



0 BEGIN PGM "M120" MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-10	
2 BLK FORM 0.2 X+110 Y+80 Z+0	; Workpiece blank definition
3 TOOL CALL 6 Z S1000 F1000	; Insert the tool with diameter 12
4 L X-5 Y+26 R0 FMAX M3	; Move in the working plane
5 L Z-5 R0 FMAX	; Infeed in the tool axis
6 L X+0 Y+20 RL F AUTO M120 LA5	; Activate contour pre-calculation and move to the first contour point
7 L X+40 Y+30	
8 CR X+47 Y+31 R-5 DR+	
9 L X+80 Y+50	
10 L X+80 Y+45	
11 L X+110 Y+45	; Move to the last contour point
12 L Z+100 R0 FMAX M120	; Retract the tool and reset **M120**
13 M30	; End of program
14 END PGM "M120" MM	

Definition

Abbreviation	Definition
LA (look ahead)	Number of look-ahead blocks

22.4.10 Shorter-path traversing of rotary axes with M126

Application

With **M126** the control moves a rotary axis on the shortest path of traverse to the programmed coordinates. This function is effective only for rotary axes whose position display is reduced to a value of less than 360°.

Description of function

Effect

M126 takes effect at the start of the block.

In order to reset **M126**, program **M127**.

Application example

11 L C+350	; Move in the C axis
12 L C+10 M126	; Shortest-path traverse in the C axis

In the first NC block the control positions the C axis to 350°.

In the second NC block the control activates **M126** and then positions the C axis with shortest-path traverse to 10°. The control uses the shortest traverse path and moves the C axis in the positive direction of rotation, beyond 360°. The traverse path is 20°.

Without **M126** the control does not move the rotary axis beyond 360°. The traverse path is then 340° in the negative direction of rotation.

Notes

- **M126** is not in effect with incremental traverse movements.
- The effect of **M126** depends on the configuration of the rotary axis.
- **M126** has an effect only on modulo axes.

  In the machine parameter **isModulo** (no. 300102) the machine manufacturer defines whether a rotary axis is a modulo axis.
- In the optional machine parameter **shortestDistance** (no. 300401) the machine manufacturer defines whether the control by default positions the rotary axis on the shortest traverse path.
- In the optional machine parameter **startPosToModulo** (no. 300402) the machine manufacturer defines whether the control reduces the actual position display to a range between 0° and 360° before each positioning.

Definitions

Modulo axis
Modulo axes are axes whose encoder only returns values between 0° and 359.9999°. If an axis is used as a spindle, then the machine manufacturer must configure this axis as a modulo axis.

Rollover axis
Rollover axes are rotary axes that can perform several or any number of revolutions. The machine manufacturer must configure a rollover axis as a modulo axis.

Modulo counting method
The position display of a rotary axis with the modulo counting method is between 0° and 359.9999°. If the value exceeds 359.9999°, the display starts over at 0°.

22.4.11 Automatically compensating for tool inclination with M128 (option 9)

Application

If the position of a controlled rotary axis changes in the NC program, then the control uses **M128** during the tilting procedure to automatically compensate for the tool inclination with a compensating movement of the linear axes. That way the position of the tool tip relative to the workpiece surface remains unchanged (TCPM).

> Instead of **M128**, HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM**.

Related topics

- Compensating for tool offset with **FUNCTION TCPM**

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027

Requirement

- Machine with rotary axes
- Kinematics description

  > Refer to your machine manual.
  >
  > The machine manufacturer creates the kinematics description of the machine.

- Software option 9: Advanced Functions (set 2)

Description of function

Effect

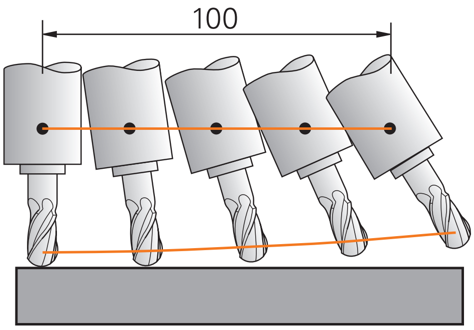
M128 takes effect at the start of the block.

You can reset **M128** with the following functions:

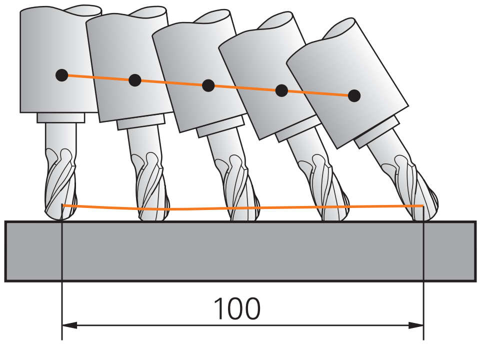
- **M129**
- **FUNCTION RESET TCPM**
- In the **Program Run** operating mode, select a different NC program

> **M128** is also in effect in the **Manual** operating mode and remains active even after a change in the operating mode.

Application example



Behavior without **M128**



Behavior with **M128**

11 L X+100 B-30 F800 M128 F1000	; Move with automatic compensation of the motion in the rotary axis

In this NC block the control activates **M128** with the feed rate for the compensating movement. The control then simultaneously moves the tool in the X axis and in the B axis.

In order to keep the position of the tool tip constant relative to the workpiece while inclining the rotary axis, the control uses the linear axes to perform a continuous compensating movement. In this example the control performs the compensating movement in the Z axis.

Without **M128** an offset of the tool tip relative to the nominal position results as soon as the inclination angle of the tool changes. The control does not compensate for this offset. If you do not take this deviation into account in the NC program, the machining operation will not be performed correctly or a collision will occur.

Input

If you define **M128**, the control continues the dialog and prompts you for the feed rate **F**. The defined value limits the feed rate during the compensating movement.

Inclined machining with open-loop rotary axes

With open-loop rotary axes, also known as counter axes, you can also perform inclined machining in combination with **M128**.

For inclined machining operations with open-loop rotary axes, proceed as follows:

- ▶ Before activating **M128**, position the rotary axes manually
- ▶ Activate **M128**
- > The control reads the actual values of all existing rotary axes, calculates from this the new position of the tool location point, and updates the position display.

  Further information: "Presets on the tool", Page 244
- > The control performs the necessary compensating movement with the next traverse movement.
- ▶ Execute the machining operation
- ▶ Reset **M128** at the program end with **M129**
- ▶ Return the rotary axes to their initial position

> ℹ As long as **M128** is active, the control monitors the actual positions of the open-loop rotary axes. If the actual position deviates from the value that is defined by the machine manufacturer, then the control issues an error message and interrupts program run.

Notes

NOTICE

Danger of collision!

Rotary axes with Hirth coupling must move out of the coupling to enable tilting. There is a danger of collision while the axis moves out of the coupling and during the tilting operation.

- Make sure to retract the tool before changing the position of the rotary axis

NOTICE

Danger of collision!

For peripheral milling, if you define the tool inclination using **LN** straight lines with tool orientation **TX**, **TY**, and **TZ**, the control autonomously calculates the required positions of the rotary axes. This can result in unexpected movements.

- Use the Simulation mode to test the NC program before execution
- Verify the NC program by slowly executing it block by block

Further information: "3D tool compensation during peripheral milling (option 9)", Page 1058

Further information: "Output with vectors", Page 1210

- The feed rate for the compensating movement remains in effect until you program a new feed rate or rescind **M128**.
- If **M128** is active, the control shows the **TCPM** symbol in the **Positions** workspace.

  Further information: "Positions workspace", Page 141
- You define the inclination angle of the tool by entering the axis positions of the rotary axes directly. This way the values refer to the machine coordinate system **M-CS**. For machines with head rotation axes the tool coordinate system **T-CS** changes. For machines with table rotary axes the workpiece coordinate system **W-CS** changes.

  Further information: "Reference systems", Page 934
- If you run the following functions while **M128** is active, then the control cancels program run and issues an error message:
  - Cutting edge radius compensation **RR/RL** in turning operations (option 50)
  - **M91**
  - **M92**
  - **M144**
  - Calling a tool with **TOOL CALL**
  - Dynamic Collision Monitoring (DCM, option 40) and at the same time **M118**
- In the optional machine parameter **maxCompFeed** (no. 201303) the machine manufacturer defines the maximum speed of compensating movements.
- In the optional machine parameter **maxAngleTolerance** (no. 205303) the machine manufacturer defines the maximum angle tolerance.
- In the optional machine parameter **maxLinearTolerance** (no. 205305) the machine manufacturer defines the maximum linear axis tolerance.
- In the optional machine parameter **manualOversize** (no. 205304) the machine manufacturer defines a manual oversize for all collision objects.

Notes on tools

If you incline a tool while machining a contour, you must use a ball-nose cutter; otherwise the tool can damage the contour.

In order to avoid damaging a contour while machining it with a ball-nose cutter, note the following:

- With **M128** the control equates the tool rotation point with the tool location point. If the tool rotation point is at the tool tip, the tool will damage the contour if the tool is inclined. Therefore the tool location point must be at the tool center point.

  Further information: "Presets on the tool", Page 244
- In order for the control to display the tool correctly in the simulation, you must define its actual length in the column **L** of the tool management.

  When calling the tool in the NC program, define the sphere radius as a negative delta value in **DL** and thus shift the tool location point to the tool center point.

  Further information: "Tool length compensation", Page 1037

  For Dynamic Collision Monitoring (DCM, option 40) it is also important that you define the tool's actual length in the tool management.

  Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 1084
- If the tool location point is at the tool center point you must modify the coordinates of the tool axis in the NC program by the value of the sphere radius.

In **FUNCTION TCPM** you can choose the tool location point and the tool rotation point separately from each other.

Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027

Definition

Abbreviation	Definition
TCPM (tool center point management)	Maintain the position of the tool location point **Further information:** "Presets on the tool", Page 244

22.4.12 Interpreting the feed rate as mm/rev with M136

Application

With **M136** the control interprets the feed rate as millimeters per revolution. The feed rate depends on the spindle speed, for example in conjunction with the turning mode (option 50).

Further information: "Switching the operating mode with FUNCTION MODE", Page 210

Description of function

Effect

M136 takes effect at the start of the block.

In order to reset **M136**, program **M137**.

Application example

11 LBL "TURN"	
12 FUNCTION MODE TURN	; Activate turning mode
13 M136	; Switch interpretation of the feed rate to mm/rev
14 LBL 0	

M136 is here located in a subprogram in which the control activates the turning mode (option 50).

With **M136** the control interprets the feed rate as millimeters per spindle revolution, which is necessary for the turning mode. The feed rate per revolution refers to the rotational speed of the workpiece spindle. The control thus moves the tool at the programmed feed rate for every rotation of the workpiece spindle.

Without **M136** the control interprets the feed rate as millimeters per minute.

Notes

- In NC programs based on inch units, **M136** is not allowed in combination with **FU** or **FZ**.
- The workpiece spindle is not permitted to be controlled when **M136** is active.
- **M136** is not possible in combination with an oriented spindle stop. The control cannot calculate the feed rate because the spindle does not rotate during an oriented spindle stop, such as when tapping.

22.4.13 Taking rotary axes into account during machining operations with M138

Application

With **M138** you define which rotary axes the control takes into account during the calculation and positioning of spatial angles. The control excludes any axes that were not defined. That way you can reduce the number of tilting possibilities and thus avoid error messages, for example on machines with three rotary axes.

M138 is in effect in combination with the following functions:

- **M128** (option 9)

  Further information: "Automatically compensating for tool inclination with M128 (option 9)", Page 1245
- **FUNCTION TCPM** (option 9)

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027
- **PLANE** functions (option 8)

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 979
- Cycle **19 WORKING PLANE** (option 8)

  Further information: "Cycle 19 WORKING PLANE (option 8)", Page 964

Description of function

Effect

M138 takes effect at the start of the block.

In order to reset **M138**, program **M138** without entering any rotary axes.

Application example

11 L Z+100 R0 FMAX M138 A C	; Define that axes **A** and **C** should be taken into account
12 PLANE SPATIAL SPA+0 SPB+90 SPC+0 MOVE FMAX	; Tilt spatial angle **SPB** by 90°

On a six-axis machine with **A**, **B**, and **C** rotary axes you must exclude one rotary axis for spatial angle operations; otherwise too many combinations are possible.

With **M138 A C** the control calculates the axis position when tilting with spatial angles only in the **A** and **C** axes. The B axis is excluded. Therefore, in NC block **12** the control positions the spatial angle **SPB+90** with the **A** and **C** axes.

Without **M138** there are too many possibilities for tilting. The control interrupts the machining process and issues an error message.

Input

If you define **M138**, the control continues the dialog and prompts you for the rotary axes to be taken into account.

11 L Z+100 R0 FMAX M138 C	; Define that the C axis should be taken into account

Notes

- With **M138** the control excludes the rotary axes only during the calculation and positioning of spatial angles. A rotary axis that has been excluded with **M138** can nevertheless be moved in a positioning block. Please note that in this case the control does not execute any compensations.
- In the optional machine parameter **parAxComp** (no. 300205) the machine manufacturer defines whether the control includes the position of the excluded axis when calculating the kinematics.

22.4.14 Retracting in the tool axis with M140

Application

With **M140** the control retracts the tool in the tool axis.

Description of function

Effect

M140 is in effect blockwise and takes effect at the start of the block.

Application example

11 LBL "SAFE"	
12 M140 MB MAX	; Retract by the maximum distance in the tool axis
13 L X+350 Y+400 R0 FMAX M91	; Approach a safe position in the working plane
14 LBL 0	

Here **M140** is in a subprogram in which the control moves the tool to a safe position.

With **M140 MB MAX** the control retracts the tool by the maximum distance in the positive direction in the tool axis. The control stops the tool before reaching a limit switch or a collision object.

In the next NC block the control moves the tool to a safe position in the working plane.

Without **M140** the control does not execute a retraction.

Input

If you define **M140**, the control continues the dialog and prompts you for the retraction distance **MB**. You can program the retraction distance as a positive or negative incremental value. With **MB MAX** the control retracts the tool in the positive direction in the tool axis before reaching a limit switch or a collision object.

After **MB** you can define a feed rate for the retraction movement. If you do not define a feed rate, the control retracts the tool at rapid traverse.

21 L Y+38.5 F125 M140 MB+50 F750	; Retract tool at feed rate of 750 mm/min by 50 mm in the positive direction of the tool axis

21 L Y+38.5 F125 M140 MB MAX	; Retract tool at rapid traverse by the maximum distance in the positive direction in the tool axis

Notes

NOTICE

Danger of collision!

The machine manufacturer has various options for configuring the Dynamic Collision Monitoring (DCM, option 40) function. Depending on the machine, the control can continue with the NC program without an error message despite the detected collision. The control stops the tool at the last position without a collision and continues the NC program from this position. This configuration of DCM results in movements that are not defined in the program. **This behavior occurs no matter whether collision monitoring is active or inactive.** There is a danger of collision during these movements!

- Refer to your machine manual.
- Check the behavior at the machine.

NOTICE

Danger of collision!

If you use **M118** to modify the position of a rotary axis with the handwheel and then execute **M140**, the control ignores the superimposed values during the retraction movement. This results in unwanted and unpredictable movements, especially when using machines with head rotation axes. There is a danger of collision during these retraction movements!

- Do not combine **M118** with **M140** when using machines with head rotation axes.

- **M140** is also effective with a tilted working plane. For machines with head rotation axes the control moves the tool in the tool coordinate system **T-CS**.

  Further information: "Tool coordinate system T-CS", Page 946
- With **M140 MB MAX** the control retracts the tool only in the positive direction in the tool axis.
- If you define a negative value for **MB**, the control retracts the tool in the negative direction in the tool axis.
- The control gleans the necessary information about the tool axis for **M140** from the tool call.
- In the optional machine parameter **moveBack** (no. 200903) the machine manufacturer defines the distance to a limit switch or a collision object upon a maximum retraction with **MB MAX**.

Definition

Abbreviation	Definition
MB (move back)	Tool axis retraction

22.4.15 Rescinding basic rotations with M143

Application

With **M143** the control resets a basic rotation as well as a 3D basic rotation, for example after machining a workpiece that needed alignment.

Description of function

Effect

M143 is in effect blockwise and takes effect at the start of the block.

Application example

11 M143	; Reset the basic rotation

In this NC block the control resets a basic rotation that had been defined in the NC program. In the active row of the preset table the control overwrites the values of the columns **SPA**, **SPB**, and **SPC** with the value **0**.

Without **M143** the basic rotation remains in effect until you manually reset the basic rotation or overwrite it with a new value.

Further information: "Preset management", Page 949

Note

The function **M143** is not permitted with mid-program startup.

Further information: "Block scan for mid-program startup", Page 1786

22.4.16 Factoring the tool offset into the calculations with M144 (option 9)

Application

The control uses **M144** in subsequent traverse movements to compensate for tool offsets that result from inclined rotary axes.

> Instead of **M144**, HEIDENHAIN recommends using the more powerful function **FUNCTION TCPM** (option 9).

Related topics

- Compensating for tool offset with **FUNCTION TCPM**

  Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027

Requirement

- Software option 9: Advanced Functions (set 2)

Description of function

Effect

M144 takes effect at the start of the block.

In order to reset **M144**, program **M145**.

Application example

11 M144	; Activate tool compensation
12 L A-40 F500	; Position the A axis
13 L X+0 Y+0 R0 FMAX	; Position the **X** and **Y** axes

With **M144** the control takes the position of the rotary axes into account in the subsequent positioning blocks.

In NC block **12** the control positions the rotary axis **A**, resulting in an offset between the tool tip and the workpiece. The control compensates for this offset mathematically.

In the next NC block the control positions the **X** and **Y** axes. When **M144** is active, the control compensates for the position of the rotary axis **A** during this movement.

Without **M144** the control does not take the offset into account, and the machining operation is performed with this offset.

Notes

Refer to your machine manual.

When working with angle heads, keep in mind that the machine geometry is defined by the machine manufacturer in a kinematics description.
If you use an angle head during machining, then you must select the correct kinematics description.

- You can use **M91** and **M92** for positioning even when **M144** is active.

  Further information: "Miscellaneous functions for coordinate entries", Page 1228
- The functions **M128** and **FUNCTION TCPM** are not permitted when **M144** is active. The control will issue an error message if you try to active these functions.
- **M144** does not work in connection with **PLANE** functions. If both functions are active, then the **PLANE** function is in effect.

  Further information: "Tilting the working plane with PLANE functions (option 8)", Page 979

  With **M144** the control moves according to the workpiece coordinate system **W-CS**.

  If you activate **PLANE** functions, the control moves according to the working plane coordinate system **WPL-CS**.

  Further information: "Reference systems", Page 934

Notes In conjunction with the turning operation (option 50)

- If the inclined axis is a tilting table, the control orients the tool coordinate system **W-CS**.

  If the inclined axis is a swivel head, the control does not orient the **W-CS**.
- After inclining the rotary axis, you may have to again pre-position the turning tool in the Y coordinate and orient the position of the tool tip with Cycle **800 ADJUST XZ SYSTEM**.

  Further information: "Cycle 800 ADJUST XZ SYSTEM ", Page 681

22.4.17 Automatically lifting off upon an NC stop or a power failure with M148

Application

With **M148** the control automatically retracts the tool from the workpiece in the following situations:

- Manually triggered NC stop
- NC stop triggered by the software, for example if an error has occurred in the drive system
- Power interruption

> Instead of **M148**, HEIDENHAIN recommends using the more powerful function **FUNCTION LIFTOFF**.

Related topics

- Automatic retraction with **FUNCTION LIFTOFF**

  Further information: "Automatic tool liftoff with FUNCTION LIFTOFF", Page 1110

Requirement

- **LIFTOFF** column in the tool management

  You must define the value **Y** in the **LIFTOFF** column of the tool management.

  Further information: "Tool management ", Page 270

Description of function

Effect

M148 takes effect at the start of the block.

You can reset **M148** with the following functions:

- **M149**
- **FUNCTION LIFTOFF RESET**

Application example

11 M148	; Activate automatic retraction

This NC block activates **M148**. If an NC stop is triggered during machining, the tool is retracted by up to 2 mm in the positive direction in the tool axis. This avoids possible damage due to the tool or workpiece.

Without **M148** the axes come to a stop upon an NC stop, meaning that the tool remains at the workpiece, which might result in surfaces blemishes on the workpiece.

Notes

- When lifting the tool off with **M148**, the control will not necessarily lift it off in the tool axis direction.

  The control uses the **M149** function to deactivate the **FUNCTION LIFTOFF** function without resetting the lift-off direction. If you program **M148**, the control will activate the automatic lift-off of the tool in the lift-off direction defined by the **FUNCTION LIFTOFF** function.
- Please note that for some tools, such as side milling cutters, automatic retraction does not make sense.
- In machine parameter **on** (no. 201401), the machine manufacturer defines whether automatic lift-off is active.
- In machine parameter **distance** (no. 201402), the machine manufacturer defines the maximum lift-off height.

22.4.18 Preventing rounding off of outside corners with M197

Application

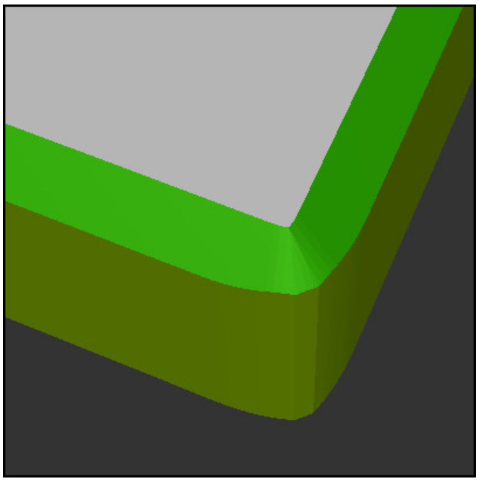
With **M197** the control extends a radius-compensated contour at the corner tangentially and inserts a smaller transition arc. That way you prevent the tool from rounding off the outside corner.

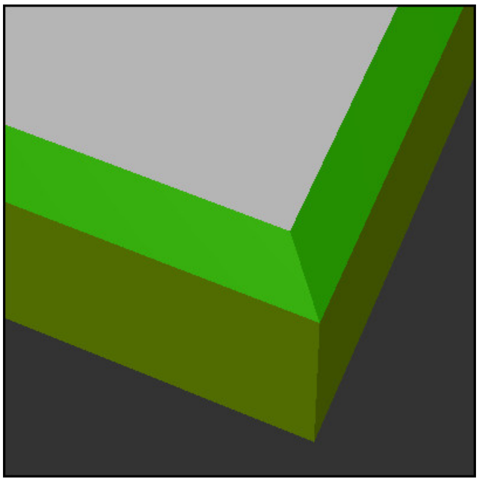
Description of function

Effect

M197 is in effect blockwise and only for radius-compensated outside corners.

Application example


Contour without **M197**


Contour with **M197**

*** - ...**	; Approach the contour
11 X+60 Y+10 M197 DL5	; Machine the first contour with a sharp edge
12 X+10 Y+60 M197 DL5	; Machine the second contour with a sharp edge
*** - ...**	; Machine the remaining contour

With **M197 DL5** the control extends the contour at the corner tangentially by up to 5 mm. In this example, the 5 mm exactly correspond to the tool radius, resulting in an outside corner with a sharp edge. The control uses the smaller transitional arc to nevertheless move along the traverse path gently.

Without **M197** and with active radius compensation the control inserts a tangential transitional arc at an outside corner, which leads to rounding off of the outside corner.

Input

If you define **M197**, the control continues the dialog and prompts you for the tangential extension **DL**. **DL** is the maximum length by which the control extends the outside corner.

Note

In order to produce corners with sharp edges, define the parameter **DL** with the same size as the tool radius. The smaller the value you enter for **DL**, the more the corner will be rounded off.

Definition

Abbreviation	Definition
DL	Maximum tangential extension

22.5 Miscellaneous functions for tools

22.5.1 Automatically inserting a replacement tool with M101

Application

With **M101** the control automatically inserts a replacement tool after a specified tool life has expired. The control then continues the machining operation with the replacement tool.

Requirements

- **RT** column in the tool management
  You define the number or name of the replacement tool in the **RT** column.
- **TIME2** column in the tool management
  In the **TIME2** column you define the tool life after which the control inserts the replacement tool.

Further information: "Tool management ", Page 270

Use only tools with an identical radius as replacement tools. The control does not automatically check the radius of the tool.

If you want the control to check the radius, program **M108** after the tool change.

Further information: "Checking the radius of the replacement tool with M108", Page 1264

Description of function

Effect

M101 takes effect at the start of the block.

In order to reset **M101**, program **M102**.

Application example

Refer to your machine manual.

The function of **M101** can vary depending on the individual machine tool.

11 TOOL CALL 5 Z S3000	; Tool call
12 M101	; Activate automatic tool change

The control exchanges the tools and activates **M101** in the next NC block. The **TIME2** column of the tool management contains the maximum age for the tool life at the time the tool is called. If, during machining, the current tool age in the column **CUR_TIME** exceeds this value, the control inserts the replacement tool at a suitable point in the NC program. This exchange takes place after no more than one minute, unless the control has not concluded the active NC block yet. A useful application of this function is for automated programs on unattended machines.

Input

If you define **M101**, the control continues the dialog and prompts you for **BT**. With **BT** you define the number of NC blocks by which the automatic tool change may be delayed (up to 100 blocks). The content of the NC blocks, such as the feed rate or distance moved, influences the time by which the tool change is delayed.

If you do not define **BT**, the control uses the value 1 or, if applicable, a default value defined by the machine manufacturer.

The value for **BT**, the tool life verification, and the calculation of the automatic tool change have an influence on the machining time.

11 M101 BT10	; Activate automatic tool change after no more than 10 NC blocks

Notes

NOTICE

Danger of collision!

During an automatic tool change with **M101**, the control always retracts the tool in the tool axis first. There is danger of collision when retracting tools for machining undercuts, such as side milling cutters or T-slot milling cutters!

- Use **M101** only for machining operations without undercuts
- Deactivate the tool change with **M102**

- If you want to reset the current age of a tool (e.g. after changing the indexable inserts), enter the value 0 in the **CUR_TIME** column of the tool management.

  Further information: "Tool management ", Page 270
- For indexed tools, the control does not apply any data from the main tool. You must define a replacement tool (with index, if necessary) in each table row in the tool management. If an indexed tool is worn and therefore disabled, this does not apply to all indices. This means, for example, that the main tool can still be used.

  Further information: "Indexed tool", Page 250
- The higher the value of **BT**, the smaller will be the effect of an extended program duration through **M101**. Please note that this will delay the automatic tool change!
- The **M101** miscellaneous function is not available for turning tools and in turning mode (option 50).

Notes on tool change

- The control performs the automatic tool change at a suitable point in the NC program.
- The control cannot perform the automatic tool change at the following points in a program.
  - During a machining cycle
  - If radius compensation with **RR** or **RL** is active
  - Directly after an **APPR** approach function
  - Directly before a **DEP** departure function
  - Directly before and after a chamfer with **CHF** or a rounding with **RND**
  - During a macro
  - During a tool change
  - Directly after the NC functions **TOOL CALL** or **TOOL DEF**
- If the machine manufacturer does not define otherwise, the control moves the tool after the tool change as follows:
  - If the target position in the tool axis is below the current position, the tool axis is positioned last.
  - If the target position in the tool axis is above the current position, the tool axis is positioned first.

Notes on the input value BT

- To calculate a suitable initial value for **BT**, use the following formula:

  $BT = 10 \div t$

  t: average machining time of an NC block in seconds

  Round the result up to an integer value. If the calculated result is greater than 100, use the maximum input value of 100.
- In the optional machine parameter **M101BlockTolerance** (no. 202206) the machine manufacturer defines the standard value for the number of NC blocks by which the automatic tool change may be delayed. This standard value applies if you do not define **BT**.

Definition

Abbreviation	Definition
BT (block tolerance)	Number of NC blocks by which a tool change may be delayed.

22.5.2 Permitting positive tool oversizes with M107 (option 9)

Application

With **M107** (option 9) the control does not interrupt the machining process upon a positive delta value. The function is in effect with active 3D tool compensation and for **LN** straight lines.

Further information: "3D tool compensation (option 9)", Page 1049

With **M107** you can, for example, use the same tool in a CAM program for pre-finishing with oversize and then later for final finishing without oversize.

Further information: "Output formats of NC programs", Page 1209

Requirement

- Software option 9: Advanced Functions (set 2)

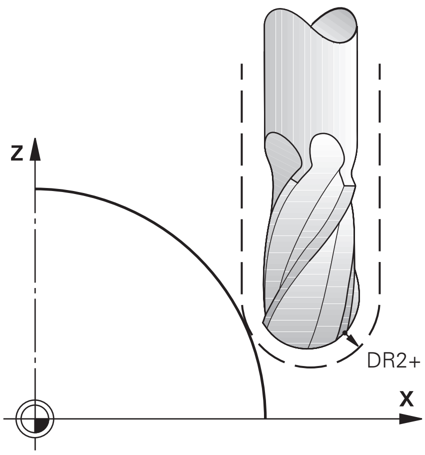
Description of function

Effect

M107 takes effect at the start of the block.

In order to reset **M107**, program **M108**.

Application example



11 TOOL CALL 1 Z S5000 DR2:+0.3	; Insert a tool with a positive delta value
12 M107	; Permit positive delta values

The control exchanges the tools and activates **M107** in the next NC block. That way the control permits positive delta values and does not issue an error message, such as during pre-finishing.

Without **M107** the control issues an error message upon positive delta values.

Notes

- Before actual machining, check in the NC program to make sure that the positive delta values of the tool will not result in contour damages or collisions.
- With peripheral milling the control issues an error message in the following case:

  $DR_{Tab} + DR_{Prog} > 0$

  Further information: "3D tool compensation during peripheral milling (option 9)", Page 1058
- With face milling the control issues an error message in the following cases:
  - $DR_{Tab} + DR_{Prog} > 0$
  - $R2 + DR2_{Tab} + DR2_{Prog} > R + DR_{Tab} + DR_{Prog}$
  - $R2 + DR2_{Tab} + DR2_{Prog} > 0$
  - $DR2_{Tab} + DR2_{Prog} > 0$

  Further information: "3D tool compensation during face milling (option 9)", Page 1053

Definition

Abbreviation	Definition
R	Tool radius
R2	Corner radius
DR	Delta value of the tool radius
DR2	Delta value of the corner radius
TAB	Value refers to the tool management
PROG	Value refers to the NC program, meaning from the tool call or from compensation tables

22.5.3 Checking the radius of the replacement tool with M108

Application

If you program **M108** before inserting a replacement tool, the control checks the replacement tool for any radius deviations.

Further information: "Automatically inserting a replacement tool with M101", Page 1260

Description of function

Effect

M108 takes effect at the end of the block.

Application example

11 TOOL CALL 1 Z S5000	; Insert the tool
12 M101 M108	; Activate automatic tool change and radius checking

The control exchanges the tool and activates the automatic tool change and radius checking in the next NC block.

If the maximum tool age of the tool expires during machining, the control inserts the replacement tool. The control checks the tool radius of the replacement tool based on the **M108** miscellaneous function defined previously. If the radius of the replacement tool is greater than the radius of the tool being replaced, the control issues an error message.

Without **M108** the control will not check the radius of the replacement tool.

Note

M108 is also used to reset **M107** (option 9).

Further information: "Permitting positive tool oversizes with M107 (option 9)", Page 1262

22.5.4 Suppressing touch probe monitoring with M141

Application

In conjunction with the touch probe cycles **3 MEASURING** or **4 MEASURING IN 3-D**, if the stylus is deflected, you can retract the touch probe in a positioning block with **M141**.

Description of function

Effect

M141 is in effect blockwise for straight lines and takes effect at the start of the block.

Application example

11 TCH PROBE 3.0 MEASURING	
12 TCH PROBE 3.1 Q1	
13 TCH PROBE 3.2 Y ANGLE: +0	
14 TCH PROBE 3.3 ABST +10 F100	
15 TCH PROBE 3.4 ERRORMODE1	
16 L IX-20 R0 F500 M141	; Retract with **M141**

In cycle **3 MEASURING** the control probes the X axis of the workpiece. Since no retraction distance **MB** is defined in this cycle, the touch probe stands still after the deflection.

In NC block **16** the control retracts the touch probe against the probing direction by 20 mm. **M141** suppresses monitoring of the touch probe.

Without **M141** the control issues an error message as soon as you move the machine axes.

Further information: "Cycle 3 MEASURING ", Page 1663

Further information: "Cycle 4 MEASURING IN 3-D ", Page 1665

Note

NOTICE

Danger of collision!

The miscellaneous function **M141** suppresses the corresponding error message if the stylus is deflected. The control does not perform an automatic collision check with the stylus. Based on these two types of behavior, you must check whether the touch probe can retract safely. There is a risk of collision if you choose the wrong direction for retraction.

- Carefully test the NC program or program section in the **Program run, single block** operating mode

23

Variable Programming

23.1 Overview of variable programming

The control provides the following options for variable programming in the **Variables** folder of the **Insert NC function** window:

Function group	Further information
Basic arithmetic operations	Page 1278
Trigonometric functions	Page 1280
Circle calculation	Page 1282
Jump commands	Page 1283
Special functions	Page 1284 Page 1296
SQL commands	Page 1316
String functions	Page 1301
Counters	Page 1308
Entering formulas directly	Page 1298
Function for machining complex contours	Page 356

23.2 Variables: Q, QL, QR and QS parameters

23.2.1 Basics

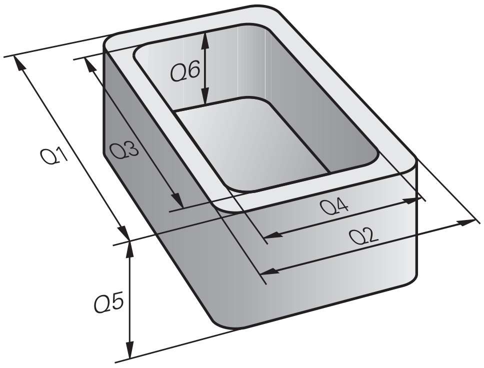
Application

You can use the Q, QL, QR and QS parameter variables of the control to perform calculations in an NC program, for example.

For instance, you can program the following syntax elements variably:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

Description of function



Q parameters are always identified with letters and numbers. The letters determine the type of Q parameter and the numbers the Q parameter range.

You can define which parameters the control displays on the **QPARA** tab of the **Status** workspace for each variable type.

Further information: "Defining the contents of the QPARA tab", Page 169

Variable types

The control provides the following variables for numerical values:

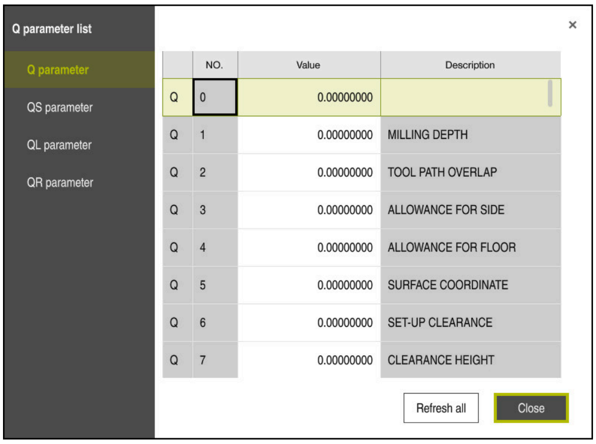
Q parameter type	Q parameter range	Meaning
Q parameters:		**Parameters affect all NC programs in the control's memory**
	0 to 99	Parameters for the **user**, if there are no overlaps with the HEIDENHAIN-SL cycles These parameters have a local effect within macros and OEM cycles. This means that changes are not returned to the NC program. For this reason, use the Q parameter range 1200 to 1399 for OEM cycles!
	100 to 199	Parameters for special functions on the control that can be read by NC programs of the user or by cycles
	200 to 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 to 1399	Parameters primarily used for OEM cycles
	1400 to 1999	Parameters for **users**
QL parameters:		**Parameters only effective locally within an NC program**
	0 to 499	Parameters for **users**
QR parameters:		**Parameters are retained in all NC programs in the control's memory, including after a power interruption**
	0 to 99	Parameters for **users**
	100 to 199	Parameters for HEIDENHAIN functions (e.g., cycles)
	200 to 499	Parameters for the machine manufacturer (e.g., cycles)

In addition, the control provides **QS** parameters for alphanumeric values, e.g. texts:

Q parameter type	Q parameter range	Meaning
QS parameters:		**Parameters affect all NC programs in the control's** memory
	0 to 99	Parameters for the **user**, where no overlaps with the HEIDENHAIN SL cycles are present These parameters have a local effect within macros and OEM cycles. This means that changes are not returned to the NC program. For this reason, use the QS parameter range 200 to 499 for OEM cycles!
	100 to 199	Parameters for special functions on the control that can be read by NC programs of the user or by cycles
	200 to 1199	Parameters primarily used for HEIDENHAIN cycles
	1200 to 1399	Parameters primarily used for OEM cycles
	1400 to 1999	Parameters for **users**

Q parameter list window

In the **Q parameter list** window, you can see and edit the current values of all variables defined in the control.



Q parameter list window, showing the Q parameter values

In the left-hand panel, you can select the variable type to be displayed.

The control displays the following information:

- Variable type, e.g. Q parameter
- Number of the variable
- Value of variable
- Description in case of pre-assigned variables

If the field in the **Value** column is displayed with a white background, you can enter a value or edit the existing value.

> While the control is executing an NC program, you cannot edit the variables using the **Q parameter list** window. Changes are only possible while a program run has been interrupted or aborted.
>
> **Further information:** "Status overview on the control bar", Page 147
>
> This status is reached after an NC block has been executed, for example in the **Single Block** mode
>
> The following Q and QS parameters cannot be edited in the **Q parameter list** window:
>
> - Parameters with numbers in the range from 100 to 199, because there might be interferences with special functions in the control.
> - Parameters with numbers in the range from 1200 to 1399, because there might be interferences with machine manufacturer-specific functions.
>
> **Further information:** "Variable types", Page 1270

You can select the **NR** or **Value** columns and then enter the desired string. The control will search the selected column for this string.

You can open the **Q parameter list** window in the following operating modes:

- **Editor**
- **Manual**
- **Program Run**

In the **Manual** and **Program Run** operating modes, the window can be opened with the **Q** key.

Notes

NOTICE

Danger of collision!

HEIDENHAIN cycles, machine manufacturer cycles and third-party functions use variables. You can also program variables within NC programs. If you do not use only the recommended variable ranges when using variables, this can lead to overlaps (interactions) and therefore to undesirable behavior. Danger of collision during machining!

- Only use variable ranges recommended by HEIDENHAIN
- Comply with the documentation from HEIDENHAIN, the machine manufacturer and third-party providers
- Check the machining sequence using the simulation

- You can enter fixed and variable values mixed in the NC program.
- You can assign a maximum of 255 characters to QS parameters.
- You can use the **Q** key to create an NC block to assign a value to a variable. If you press the key again, the control changes the variable type in the order **Q**, **QL**, **QR**.

  On the virtual keyboard, this procedure only works with the **Q** key in the NC functions area.

  Further information: "Virtual keyboard of the control bar", Page 1380
- Q parameters can be assigned numerical values between -999 999 999 and +999 999 999. The input range is limited to 16 digits, of which 9 may be before the decimal point. Internally the control calculates numbers up to a value of 10^{10}.
- You can reset Q parameters to the status **Undefined**. If a position is programmed with a Q parameter that is undefined, the control ignores this movement.

  Further information: "Setting a variable to undefined", Page 1280
- You must not use preassigned Q parameters (QS parameters) between **Q100** and **Q199** (**QS100** and **QS199**) as calculation parameters in the NC programs.

  Further information: "Preassigned Q parameters", Page 1273
- The control saves numerical values internally in a binary number format (standard IEEE 754). Due to the standardized format used, the control does not represent some decimal numbers with a binary number that is 100% exact (round-off error). If you use calculated variable contents for jump commands or positioning moves, you must take this into consideration.

Notes on QR parameters and backup

The control saves QR parameters within a backup.

If the machine manufacturer did not define a specific path, the control will save the **QR** parameter values in the following path: **SYS:\runtime\sys.cfg**. This partition will only be backed up in full backups.

Machine manufacturers can use the following optional machine parameters to specify the paths:

- **pathNcQR** (no. 131201)
- **pathSimQR** (no. 131202)

If the machine manufacturer used the optional machine parameters to specify a path on the TNC partition, you can perform a backup with the **NC/PLC Backup** functions without entering a code number.

Further information: "Backup and restore", Page 1955

23.2.2 Preassigned Q parameters

The Q parameters **Q100** to **Q199** are assigned values by the control. The following types of information are assigned to the Q parameters:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Results of measurements from touch probe cycles etc.

The control saves the preassigned Q parameters **Q108** and **Q114** to **Q117** in the unit of measure used by the active NC program.

Values from the PLC: Q100 to Q107

The control assigns values from the PLC to parameters **Q100** to **Q107** in an NC program.

Active tool radius: Q108

The active value of the tool radius is assigned to **Q108**. **Q108** is calculated from:

- Tool radius **R** from the tool table
- Delta value **DR** from the tool table
- Delta value **DR** from the NC program (compensation table or tool call)

> The control remembers the current tool radius even if the power is interrupted.

Tool axis: Q109

The value of the parameter **Q109** depends on the current tool axis:

Parameter	Tool axis
Q109 = –1	No tool axis defined
Q109 = 0	X axis
Q109 = 1	Y axis
Q109 = 2	Z axis
Q109 = 6	U axis
Q109 = 7	V axis
Q109 = 8	W axis

Spindle status: Q110

The value of **Q110** depends on the last defined additional function for the spindle:

Parameter	Additional function
Q110 = –1	No spindle status defined
Q110 = 0	M3: Spindle ON, clockwise
Q110 = 1	M4: Spindle ON, counterclockwise
Q110 = 2	M5 after M3
Q110 = 3	M5 after M4

Coolant on/off: Q111

Parameter	M function
Q111 = 1	M8: Coolant ON
Q111 = 0	M9: Coolant OFF

Overlap factor: Q112

The control assigns **Q112** to the overlap factor for pocket milling.

Unit of measurement for dimensions in the NC program: Q113

During nesting with **PGM CALL**, the value of the parameter **Q113** depends on the dimensional data of the NC program from which the other NC programs are called.

Parameter	Dimensional data of the main program
Q113 = 0	Metric system (mm)
Q113 = 1	Imperial system (inch)

Tool length: Q114

The current value for the tool length is assigned to **Q114**.

The control remembers the current tool length even if the power is interrupted.

Coordinates after probing during program run

The parameters **Q115** to **Q119** contain the coordinates of the spindle position at the instant of probing during programmed measurement with the 3D touch probe. The coordinates are referenced to the preset that is active in **Manual operation** mode.

The length of the stylus and the radius of the ball tip are not compensated in these coordinates.

Parameter	Coordinate axis
Q115	X axis
Q116	Y axis
Q117	Z axis
Q118	4th axis Machine-dependent
Q119	5th axis Machine-dependent

Deviation between actual and nominal value during automatic tool measurement; for example, with the TT 160

Parameter	Deviation of actual from nominal value
Q115	Tool length
Q116	Tool radius

Tilting the working plane with workpiece angles: Coordinates calculated by the control for rotary axes

Parameter	Coordinates
Q120	A axis
Q121	B axis
Q122	C axis

Measurement results from touch probe cycles

Parameters	Measured actual values
Q150	Angle of a straight line
Q151	Center in main axis
Q152	Center in secondary axis
Q153	Diameter
Q154	Pocket length
Q155	Pocket width
Q156	Length of the axis selected in the cycle
Q157	Position of the centerline
Q158	Angle in the A axis
Q159	Angle in the B axis
Q160	Coordinate of the axis selected in the cycle

Parameters	Measured deviation
Q161	Center in main axis
Q162	Center in secondary axis
Q163	Diameter
Q164	Pocket length
Q165	Pocket width
Q166	Measured length
Q167	Position of the centerline

Parameters	Determined space angle
Q170	Rotation about the A axis
Q171	Rotation about the B axis
Q172	Rotation about the C axis

Parameters	Workpiece status
Q180	Good
Q181	Rework
Q182	Scrap

Parameters	Tool measurement with the BLUM laser
Q190	Reserved
Q191	Reserved
Q192	Reserved
Q193	Reserved

Parameters	Reserved for internal use
Q195	Marker for cycles
Q196	Marker for cycles

Parameters	Reserved for internal use
Q197	Marker for cycles (machining patterns)
Q198	Number of the last active measuring cycle

Parameter value	Status of tool measurement with TT
Q199 = 0.0	Tool is within the tolerance.
Q199 = 1.0	Tool is worn (LTOL/RTOL is exceeded)
Q199 = 2.0	Tool is broken (LBREAK/RBREAK is exceeded)

Parameters	Measured actual values
Q950	1st position in the main axis
Q951	1st position in the secondary axis
Q952	1st position in the tool axis
Q953	2nd position in the main axis
Q954	2nd position in the secondary axis
Q955	2nd position in the tool axis
Q956	3rd position in the main axis
Q957	3rd position in the secondary axis
Q958	3rd position in the tool axis
Q961	Spatial angle SPA in the WPL-CS
Q962	Spatial angle SPB in the WPL-CS
Q963	Spatial angle SPC in the WPL-CS
Q964	Angle of rotation in the I-CS
Q965	Angle of rotation in the coordinate system of the rotary table
Q966	First diameter
Q967	Second diameter

Parameters	Measured deviations
Q980	1st position in the main axis
Q981	1st position in the secondary axis
Q982	1st position in the tool axis
Q983	2nd position in the main axis
Q984	2nd position in the secondary axis
Q985	2nd position in the tool axis
Q986	3rd position in the main axis
Q987	3rd position in the secondary axis
Q988	3rd position in the tool axis
Q994	Angle in the I-CS
Q995	Angle in the coordinate system of the rotary table
Q996	First diameter
Q997	Second diameter

Parameter value	Workpiece status
Q183 = -1	Not defined
Q183 = 0	Pass
Q183 = 1	Rework
Q183 = 2	Scrap

23.2.3 Basic arithmetic folder

Application

In the **Basic arithmetic** folder of the **Insert NC function** window, the control offers the functions **FN 0** to **FN 5**.

You can assign numeric values to variables using the **FN 0** function. You then use a variable instead of the fixed number in the NC program. You can also use pre-assigned variables, e.g. results from touch probe cycles. With the functions **FN 1** to **FN 5**, you can make calculations with the variable values within the NC program.

Related topics

- Preassigned variables

  Further information: "Preassigned Q parameters", Page 1273
- Programmable touch probe cycles

  Further information: "Programmable Touch Probe Cycles", Page 1449
- Calculations with multiple calculation steps in an NC block

  Further information: "Formulas in the NC program", Page 1298

Description of function

The **Basic arithmetic** folder contains the following functions:

Icon	Function
=	**FN 0**: ASSIGN e. g., **FN 0: Q5 = +60** Directly assign value Reset Q parameter value
+	**FN 1**: ADDITION e. g., **FN 1: Q1 = -Q2 + -5** Calculate and assign the sum of two values
−	**FN 2**: SUBTRACTION e. g. **FN 2: Q1 = +10 - +5** Form and assign difference between two values
×	**FN 3**: MULTIPLICATION e. g. **FN 3: Q2 = +3 * +3** Form and assign the product of two values
∕	**FN 4**: DIVISION e.g., **FN 4: Q4 = +8 DIV +Q2** Calculate and assign the quotient of two values **Not permitted:** Division by 0
√	**FN 5**: SQUARE ROOT e.g., **FN 5: Q20 = SQRT 4** Calculate and assign the square root of a value **Not permitted:** Square root of a negative value

To the left of the **=** sign, you define the variable to which you assign the value or result.

To the right of the **=** sign, you can use fixed or variable values.

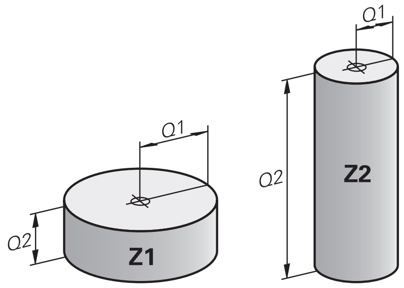
The variables and numerical values in the equations can be entered with an algebraic sign.

Part families

For part families, you program the characteristic workpiece dimensions as variables, for example. To process the individual parts, assign a corresponding numerical value to each variable.

11 LBL "Z1"	
12 FN 0: Q1=30	; Assignment, **Q1** contains the value 30
13 FN 0: Q2=10	; Assignment, **Q2** contains the value 10
*** - ...**	
21 L X +Q1	; Corresponds to **L** X +30

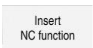
Example: Cylinder with Q parameters



Cylinder radius:	R = Q1
Cylinder height:	H = Q2
Cylinder Z1:	Q1 = +30 Q2 = +10
Cylinder Z2:	Q1 = +10 Q2 = +50

Setting a variable to undefined

To set a variable to the **undefined** status:



- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **FN 0**
- ▶ Enter the number of the variable, e.g. **Q5**
- ▶ Select **SET UNDEFINED**
- ▶ Confirm your input
- > The control sets the variable to the **undefined** status.

Notes

- The control distinguishes between undefined variables and variables with the value 0.
- You cannot divide by 0 (**FN 4**).
- You cannot extract a square root from a negative value (**FN 5**).

23.2.4 Trigonometric functions folder

Application

In the **Trigonometric functions** folder of the **Insert NC function** window, the control provides the functions **FN 6** to **FN 8** and **FN 13**.

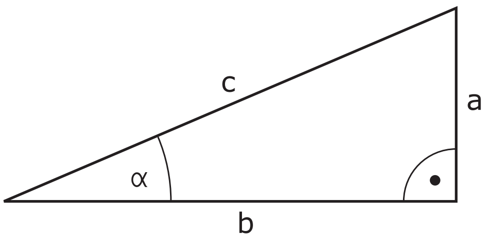
You can use these functions to calculate trigonometric functions for purposes such as programming variable triangular contours.

Description of function

The **Trigonometric functions** folder contains the following functions:

Icon	Function
SIN	**FN 6**: SINE e. g., **FN 6: Q20 = SIN-Q5** Calculate and assign the sine of an angle in degrees (°)
COS	**FN 7**: COSINE e. g., **FN 7: Q21 = COS-Q5** Calculate and assign the cosine of an angle in degrees (°)
LEN	**FN 8**: ROOT SUM OF SQUARES e. g., **FN 8: Q10 = +5 LEN +4** Calculate and assign length from two values
ANG	**FN 13**: ANGLE e. g., **FN 13: Q20 = +25 ANG-Q1** Calculate and assign an angle with the arc tangent of the opposite and adjacent sides of the triangle or with the sine and cosine of the angle (0 < angle < 360°).

Definition



Side or trigonometric function	Meaning
a	Side opposite angle **α**
b	Third side
c	Side opposite right angle
Sine	sin **α** = a / c
Cosine	cos **α** = b / c
Tangent	tan **α** = a / b = sin **α** / cos **α**
Arc tangent	**α** = arctan (a / b) = arctan (sin **α** / cos **α**)

Example

a = 25 mm

b = 50 mm

α = arctan (a / b) = arctan 0.5 = 26.57°

Furthermore:

$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

$c = \sqrt{(a^2 + b^2)}$

23.2.5 Circle calculation folder

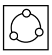
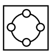
Application

In the **Circle calculation** folder of the **Insert NC function** window, the control provides the functions **FN 23** and **FN 24**.

These functions allow you to calculate the center of the circle and the radius of the circle from the coordinates of three or four circle points, for example the position and size of a partial circle.

Description of function

The **Circle calculation** folder contains the following functions:

Icon	Function
	FN 23: Determining the CIRCLE DATA from three points Example: **FN 23: Q20 = CDATA Q30**
	FN 24: Determining the CIRCLE DATA from four points Example: **FN 24: Q20 = CDATA Q30**

You save the coordinates in the working plane of the respective points in consecutive variables. You must save the coordinates of the main axis before the coordinates of the secondary axis, e.g. **X** before **Y** for tool axis **Z**.

Further information: "Designation of the axes on milling machines", Page 186

Circle calculation with three circle points

```
11 FN 23: Q20 = CDATA Q30
```

The coordinate pairs of three points on a circle must be saved in **Q30** and the five subsequent parameters—in this case, up to **Q35**.

The control then saves the circle center in the principal axis (X if spindle axis is Z) in parameter **Q20**, the circle center in the minor axis (Y if spindle axis is Z) in parameter **Q21**, and the circle radius in parameter **Q22**.

Circle calculation with four circle points

```
11 FN 24: Q20 = CDATA Q30
```

The coordinate pairs of four points on a circle must be saved in parameter **Q30** and the seven subsequent parameters—in this case, up to **Q37**.

The control then saves the circle center in the principal axis (X if spindle axis is Z) in parameter **Q20**, the circle center in the minor axis (Y if spindle axis is Z) in parameter **Q21**, and the circle radius in parameter **Q22**.

Note

Note that **FN 23** and **FN 24** automatically overwrite the resulting parameter and the two following parameters.

23.2.6 Jump commands folder

Application

In the **Jump commands** folder of the **Insert NC function** window, the control provides the functions **FN 9** to **FN 12** for jumps with if-then decisions.

In if-then decisions, the control compares a Q parameter with another Q parameter or a numerical value. If the condition is fulfilled, then the control continues the NC program at the label that is programmed after the condition.

If the condition is not fulfilled, the control continues with the next NC block.

Related topics

- Jumps without condition with **CALL LBL** label call

  Further information: "Subprograms and program section repeats with the label LBL", Page 332

Description of function

The **Jump commands** folder contains the following functions for if-then decisions:

Icon	Function
=	**FN 9**: IF EQUAL, JUMP e. g., **FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25"** If both values or parameters are equal, jump to specified label
	FN 9: IF UNDEFINED, JUMP e. g., **FN 9: IF +Q1 IS UNDEFINED GOTO LBL “UPCAN25“** If the specified parameter is undefined, then a jump is made to the specified label
	FN 9: IF DEFINED, JUMP e. g., **FN 9: IF +Q1 IS DEFINED GOTO LBL “UPCAN25“** If the specified parameter is defined, then a jump is made to the specified label
≠	**FN 10**: IF UNEQUAL, JUMP e. g., **FN 10: IF +10 NE -Q5 GOTO LBL 10** If both values or parameters are unequal, jump to specified label
>	**FN 11**: IF GREATER, JUMP g. g., **FN 11: IF+Q1 GT+10 GOTO LBL QS5** If the first value or parameter is greater than the second value or parameter, jump to specified label
<	**FN 12**: IF LESS, JUMP e. g., **FN 12: IF+Q5 LT+0 GOTO LBL "ANYNAME"** If the first value or parameter is smaller than the second value or parameter, jump to specified label

For the **IF** condition, you can enter fixed or variable numbers or text.

For the jump address **GOTO**, you can enter the following values:

- **LBL NAME**
- **LBL NUMBER**
- **QS**

Unconditional jump

Unconditional jumps are jumps whose condition is always fulfilled.

Example:

```
11 FN 9: IF+0 EQU+0 GOTO LBL1
```

You can use such jumps, for example, in a called NC program in which you work with subprograms. This way, you can ensure that the control does not execute the subprograms without a call, even without **M30** or **M2**.

Further information: "Subprograms", Page 334

Definitions

Abbreviation	Definition
IF	If
EQU (equal)	Equal to
NE (not equal)	Not equal to
GT (greater than)	Greater than
LT (less than)	Less than
GOTO (go to)	Go to
UNDEFINED	Undefined
DEFINED	Defined

23.2.7 Special functions for programming with variables

Output error messages with FN 14: ERROR

Application

With the **FN 14: ERROR** error function, you can output error messages under program control. The messages are predefined by the machine manufacturer or by HEIDENHAIN.

Related topics

- Error numbers pre-assigned by HEIDENHAIN

  Further information: "Preassigned error numbers for FN 14: ERROR", Page 2040
- Error messages in the notification menu

  Further information: "Message menu on the information bar", Page 1400

Description of function

If, during a program run or during simulation, the control encounters the **FN 14: ERROR** function, it will interrupt the program run and display the defined message. You must then restart the NC program.

You define the error number for the desired error message.

The error numbers are grouped as follows:

Error number range	Standard dialog
0 ... 999	Machine-dependent dialog
1000 ... 1199	Internal error messages

Note

Note that depending on the type of control you have, not all error messages are present.

Outputting text formatted with FN 16: F-PRINT

Application

With the function **FN 16: F-PRINT**, you can save Q parameter values and output formatted texts (e.g. in order to save measurement reports).

You can output the values as follows:

- Save them to a file on the control
- Display them on the screen in a pop-up window
- Save them to an external file
- Print them using a connected printer

Related topics

- Automatically generated measurement log for touch probe cycles
  Further information: "Recording the results of measurement", Page 1607
- Print to a connected printer
  Further information: "Printers", Page 1937

Description of function

To output Q-parameter values and texts:

- Create a text file that defines the output format and contents
- In the NC program, use the function **FN 16: F-PRINT** in order to output the log

If you output the values to a file, the maximum size of the output file will be 20 KB.

Within the **FN 16** function, you specify the output file that contains the texts to be output.

The control displays the output file in the following cases:

- End of program **END PGM**
- Cancellation of program with the **NC STOP** key
- **M_CLOSE** command
  Further information: "Keywords", Page 1287

Source file for output format

You define the output format and the contents of the file in a source file ***.a**.

Formatting

You can define the formatting using the following commands:

> Pay attention to capitalization.

Special characters	Function
"..........."	Define output format for texts and variables between the quotation marks
%F	Format for Q parameters, QL, and QR: ■ Define %: format ■ F: Floating (decimal number), format for Q, QL, QR
9.3	Format for Q parameters, QL, and QR: ■ Total of 9 characters, including decimal separator ■ Of these, 3 are decimal places
%S	Format for text variable QS
%RS	Format for text variable QS Assumes the subsequent without any changes or formatting
%D or **%I**	Format for integer
,	Separation character between output format and parameter
;	End of block character
*****	Beginning of a comment line Comments are not shown in the log
%"	Output quotation marks
%%	Output percent sign
****	Output backslash
\n	Output line break
+	Q parameter value, right-aligned
-	Q parameter value, left-aligned

Keywords

You can include the following information in the file:

Keyword	Function
CALL_PATH	Gives the path for the NC program where you will find the FN 16 function. Example: "Measuring program: %S",CALL_PATH;
M_CLOSE	Closes the file to which you are writing with FN 16. Example: M_CLOSE;
M_APPEND	Upon renewed output, appends the log to the existing log. Example: M_APPEND;
M_APPEND_MAX	Upon renewed output, appends the log to the existing log until the maximum specified file size in kilobytes is exceeded. Example: M_APPEND_MAX20;
M_TRUNCATE	Overwrites the log upon renewed output. Example: M_TRUNCATE;
M_EMPTY_HIDE	Prevents empty lines from being inserted into the log if QS parameters are not defined or empty. Example: M_EMPTY_HIDE;
M_EMPTY_SHOW	Inserts empty lines into the log if QS parameters are not defined. Resets M_EMPTY_HIDE. Example: M_EMPTY_SHOW;
L_ENGLISH	Outputs the text only for English conversational language
L_GERMAN	Outputs the text only for German conversational language
L_CZECH	Outputs text only for Czech conversational language
L_FRENCH	Outputs text only for French conversational language
L_ITALIAN	Outputs text only for Italian conversational language
L_SPANISH	Outputs text only for Spanish conversational language
L_PORTUGUE	Outputs text only for Portuguese conversational language
L_SWEDISH	Outputs text only for Swedish conversational language
L_DANISH	Outputs text only for Danish conversational language
L_FINNISH	Outputs text only for Finnish conversational language
L_DUTCH	Outputs text only for Dutch conversational language
L_POLISH	Outputs text only for Polish conversational language
L_HUNGARIA	Outputs text only for Hungarian conversational language
L_CHINESE	Outputs text only for Chinese conversational language
L_CHINESE_TRAD	Outputs text only for Chinese (traditional) conversational language
L_SLOVENIAN	Outputs text only for Slovenian conversational language
L_NORWEGIAN	Outputs text only for Norwegian conversational language

Keyword	Function
L_ROMANIAN	Outputs text only for Romanian conversational language
L_SLOVAK	Outputs text only for Slovakian conversational language
L_TURKISH	Outputs text only for Turkish conversational language
L_ALL	Display text independently of the conversational language
HOUR	Number of hours from the real-time clock
MIN	Number of minutes from the real-time clock
SEC	Number of seconds from the real-time clock
DAY	Day from the real-time clock
MONTH	Month as a number from the real-time clock
STR_MONTH	Month as a string abbreviation from the real-time clock
YEAR2	Two-digit year from the real-time clock
YEAR4	Four-digit year from the real-time clock

Input

11 FN 16: F-PRINT TNC:\mask.a / TNC:\Prot1.txt	; Output file **Prot1.txt** with the source from **Mask.a**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 16: F-PRINT	Syntax initiator for outputting formatted texts
***.a**	Path of the source file for the output format
/	Separator between the two paths
TNC:\Prot1.txt	Path under which the control saves the output file The file name extension of the log file determines the file type of the output (e.g., TXT, A, XLS, HTML).

You can enter the source file and the output file as Q parameters or as QS parameters. For this purpose you previously define the desired parameter in the NC program.

Enter Q parameters in the **FN 16** function with the following syntax so that the control can detect the Q parameters:

Input	Function
:'QS1'	Set QS parameters with preceding colon and between single quotation marks
:'QL3'.txt	Specify additional file name extension for the target file if required

Output options

Screen output

You can use the **FN 16: F-PRINT** function to display messages in a pop-up window on the control screen. This makes it easy to display explanatory texts in such a way that the user cannot continue without reacting to them. The length and position of these texts can be defined freely in the NC program. It is also possible to display the contents of variables by defining the text file accordingly.

In order to display the message on the control screen, enter **SCREEN:** as the output path.

Example

```
11 FN 16: F-PRINT TNC:\MASKE\MASKE1.A / SCREEN:
```

> If you want to replace the content of the pop-up window for multiple screen outputs in the NC program, define the **M_CLOSE** or **M_TRUNCATE** keywords.

You can close the pop-up window in the following ways:

- By pressing the **CE** key
- By defining the **SCLR:** output path

Save output externally of the control

With the **FN 16** function you can also store log files externally.

To do so you must enter the target path in the **FN 16** function.

Example

```
96 FN 16: F-PRINT TNC:\MSK\MSK1.A / PC325:\LOG\PRO1.TXT
```

If you output the same file more than once in the NC program, the control appends the current output to the end of the contents already output within the target file.

Print output file

You can also use the function **FN 16: F-PRINT** to print the output files on a connected printer.

Further information: "Printers", Page 1937

The control will only print the log file if the source file for the output format ends with the **M_CLOSE** keyword.

To ensure that the messages will be sent to the default printer, you must enter **Printer:** and a file name as the target (output path).

If you do not use the default printer, enter the path to the respective printer, e.g., **Printer:\PR0739** and a file name.

The control will save the file using the defined file name and the defined path. The file name will not be printed.

The control saves the file only until printing is complete.

Example

```
11 FN 16: F-PRINT TNC:\MASKE\MASKE1.A / PRINTER:\PRINT1
```

Notes

- Use the optional machine parameters **fn16DefaultPath** (no. 102202) and **fn16DefaultPathSim** (no. 102203) to define a path under which the control saves the output files.
- If you only define the file name as the destination path of the output file, the control saves the output file in the folder of the NC program.
- If you define a path both in the machine parameters and in the **FN 16** function, the path in the **FN 16** function has priority.
- If the called file is located in the same directory as the file you are calling it from, you can also enter just the file name without the path. If you select the file using the selection menu, the control automatically proceeds in this manner.
- With the **%RS** function in the source file, the control accepts the following content unformatted. This allows you to output a path specification with QS parameters, for example.
- For text output, you can use the UTF-8 character set.

Example

Example of a text file that outputs a log file of variable length:

"MEASURING LOG";

“%S“,QS1;

M_EMPTY_HIDE;

“%S“,QS2;

“%S“,QS3;

M_EMPTY_SHOW;

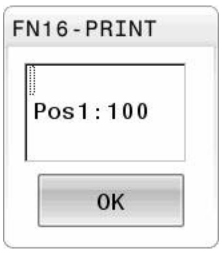
“%S“,QS4;

M_CLOSE;

Example of an NC program that defines only **QS3**:

```
95 Q1 = 100
96 QS3 = "Pos 1: " || TOCHAR( DAT+Q1 )
97 FN 16: F-PRINT TNC:\fn16.a / SCREEN:
```

Example of a screen output with two empty lines resulting from **QS1** and **QS4**:



Read system data with FN 18: SYSREAD

Application

The **FN 18: SYSREAD** function can be used to read system data and store this data in variables.

Related topics

- List of the system data of the control

  Further information: "List of FN functions", Page 2046
- Read system data using QS parameters

  Further information: "Read system data with SYSSTR", Page 1303

Description of function

The control always outputs system data in the metric system with **FN 18: SYSREAD**, regardless of the unit of the NC program.

Input

11 FN 18: SYSREAD Q25 = ID210 NR4 IDX3	; Save the active dimension factor of the Z axis in **Q25**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN18: SYSREAD	Read the syntax initiator for system data
Q/QL/QR or **QS**	Variable in which the control stores the information
ID	Group number of the system datum
NR	System data number Optional syntax element
IDX	Index Optional syntax element
.	Sub-index for system data for tools Optional syntax element

Note

As an alternative, you can use **TABDATA READ** to read out data from the active tool table. In this case, the control will automatically convert the table values to the unit of measure used in the NC program.

Further information: "Reading table values with TABDATA READ", Page 1810

Transfer values to PLC with FN 19: PLC

Application

The **FN 19: PLC** function transfers up to two numerical values or Q parameters to the PLC.

Description of function

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. This function provides HEIDENHAIN as well as your machine manufacturer and suppliers with the ability to communicate with the PLC from an NC program. It is not recommended for the machine operator or NC programmer to use this function. There is risk of collision during the execution of the function and during the subsequent machining!

- Only use the function in consultation with HEIDENHAIN, the machine manufacturer, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and suppliers.

Synchronizing NC and PLC with FN 20: WAIT FOR

Application

With the **FN 20: WAIT FOR** function you can synchronize the NC and PLC during a program run. The NC stops machining until the condition that you have programmed in the **FN 20: WAIT FOR** block is fulfilled.

Description of function

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. This function provides HEIDENHAIN as well as your machine manufacturer and suppliers with the ability to communicate with the PLC from an NC program. It is not recommended for the machine operator or NC programmer to use this function. There is risk of collision during the execution of the function and during the subsequent machining!

- Only use the function in consultation with HEIDENHAIN, the machine manufacturer, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and suppliers.

SYNC is used whenever you read, for example, system data via **FN 18: SYSREAD** that require synchronization with real time. The control stops the look-ahead calculation and executes the following NC block only when the NC program has actually reached that NC block.

Application example

```
32 FN 20: WAIT FOR SYNC
33 FN 18: SYSREAD Q1 = ID270 NR1 IDX1
```

In this example, you stop the internal pre-calculation of the control in order to determine the current position of the X axis.

Transferring values to PLC with FN 29: PLC

Application

The **FN 29: PLC** function transfers up to eight numerical values or Q parameters to the PLC.

Description of function

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. This function provides HEIDENHAIN as well as your machine manufacturer and suppliers with the ability to communicate with the PLC from an NC program. It is not recommended for the machine operator or NC programmer to use this function. There is risk of collision during the execution of the function and during the subsequent machining!

- Only use the function in consultation with HEIDENHAIN, the machine manufacturer, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and suppliers.

Creating your own cycles with FN 37: EXPORT

Application

You need the **FN 37: EXPORT** function if you want to create your own cycles and integrate them in the control.

Description of function

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., inoperability of the control). For this reason, access to the PLC is protected by password. This function provides HEIDENHAIN as well as your machine manufacturer and suppliers with the ability to communicate with the PLC from an NC program. It is not recommended for the machine operator or NC programmer to use this function. There is risk of collision during the execution of the function and during the subsequent machining!

- Only use the function in consultation with HEIDENHAIN, the machine manufacturer, or the supplier.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and suppliers.

Sending information from the NC program with FN 38: SEND

Application

The function **FN 38: SEND** enables you to retrieve texts and Q parameter values from the NC program and write them to the log or send them to an external application, e.g. StateMonitor.

Description of function

Data is transferred via a TCP/IP connection.

For more detailed information, consult the RemoTools SDK manual.

Input

`11 FN 38: SEND /"Q-Parameter Q1: %f Q23: %f" / +Q1 / +Q23`	; Write values from **Q1** and **Q23** to the logbook

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 18: SEND	Send syntax initiator for information
/	Output text as fixed or variable text with up to max. seven placeholders for the values of the variables, e.g. **%f** **Further information:** "Source file for output format", Page 1285
/	Content of the max. seven placeholders in the output text as fixed or variable numbers Optional syntax element

Notes

- Placeholders are case-sensitive, so make sure to enter them correctly.
- To obtain **%** in the output text, enter **%%** at the desired position.

Example

Send information to StateMonitor.

With function **FN 38**, you can enter job data, among others. This requires a job that has been created in StateMonitor and an assignment to the machine tool being used.

> Job management with JobTerminal (option 4) is possible with StateMonitor version 1.2 or higher.

Requirements:

- Job number 1234
- Working step 1

11 FN 38: SEND /"JOB:1234_STEP:1_CREATE"	; Create job
12 FN 38: SEND /"JOB:1234_STEP:1_CREATE_ITEMNAME: HOLDER_ITEMID:123_TARGETQ:20"	; Alternatively: Create job with part name, part number and target quantity
13 FN 38: SEND /"JOB:1234_STEP:1_START"	; Start job
14 FN 38: SEND /"JOB:1234_STEP:1_PREPARATION"	; Start preparation
15 FN 38: SEND /"JOB:1234_STEP:1_PRODUCTION"	; Production
16 FN 38: SEND /"JOB:1234_STEP:1_STOP"	; Stop job
17 FN 38: SEND /"JOB:1234_STEP:1_ FINISH"	; Finish job

In addition, the workpiece quantities for the job can be reported.

With the **OK**, **S**, and **R** placeholders, you can specify whether the quantity of reported workpieces has been machined correctly or not.

The **A** and **I** placeholders allow you to define how StateMonitor interprets the response. If absolute values are transferred, then StateMonitor overwrites the previously valid values. In the case of incremental values, StateMonitor increments the quantity.

11 FN 38: SEND /"JOB:1234_STEP:1_OK_A:23"	; Actual quantity (OK) absolute
12 FN 38: SEND /"JOB:1234_STEP:1_OK_I:1"	; Actual quantity (OK) incremental
13 FN 38: SEND /"JOB:1234_STEP:1_S_A:12"	; Scrap (S) absolute
14 FN 38: SEND /"JOB:1234_STEP:1_S_I:1"	; Scrap (S) incremental
15 FN 38: SEND /"JOB:1234_STEP:1_R_A:15"	; Rework (R) absolute
16 FN 38: SEND /"JOB:1234_STEP:1_R_I:1"	; Rework (R) incremental

23.2.8 Functions for freely definable tables

Opening a freely definable table with FN 26: TABOPEN

Application

With the function **FN 26: TABOPEN** you open a freely definable table to be written to with **FN 27** or to be read from with **FN 28**.

Related topics

- Content and creation of freely definable tables

  Further information: "Freely definable tables", Page 1850
- Access to table values with low computing power

  Further information: "Table access with SQL statements", Page 1316

Description of function

Enter the path of the freely definable table. The file name must end with ***.tab**.

Note

Only one table can be opened in an NC program at any one time. A new NC block with **FN 26: TABOPEN** automatically closes the last opened table.

Writing to a freely definable table with FN 27: TABWRITE

Application

With the **FN 27: TABWRITE** function you write to the table that you previously opened with **FN 26: TABOPEN**.

Related topics

- Content and creation of freely definable tables

  Further information: "Freely definable tables", Page 1850
- Opening a freely definable table

  Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 1296

Description of function

You can define multiple column names in a **TABWRITE** block. The column names must be written between quotation marks and separated by a comma. You define in Q parameters the value that the control is to write to the respective column.

Use QS parameters if you want to write to a text field (such as column type **UPTEXT**). Use Q, QL, or QR parameters to write to numerical fields.

Notes

- The control only executes the **FN 27: TABWRITE** function in the **Program Run** operating mode.

  The **FN 18 ID992 NR16** function allows you to query the operating mode in which the NC program is running.
- If you write to more than one column in an NC block, you must save the values under successive Q parameter numbers.
- The control displays an error message if you try to write to a table cell that is locked or does not exist.

Example

You wish to write to the columns "Radius", "Depth", and "D" in line 5 of the presently opened table. The values to be written in the table are saved in the Q parameters **Q5**, **Q6**, and **Q7**.

11 Q5 = 3,75	; Define the value for the **Radius** column
12 Q6 = -5	; Define the value for the **Depth** column
13 Q7 = 7,5	; Define the value for the **D** column
14 FN 27: TABWRITE 5/"Radius,Depth,D" = Q5	; Write defined values to the table

Reading a freely definable table with FN 28: TABREAD

Application

With the **FN 28: TABREAD** function you read from the table previously opened with **FN 26: TABOPEN**.

Related topics

- Content and creation of freely definable tables
  Further information: "Freely definable tables", Page 1850
- Opening a freely definable table
  Further information: "Opening a freely definable table with FN 26: TABOPEN", Page 1296
- Writing a freely definable table
  Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 1296

Description of function

You can define, i.e. read, multiple column names in a **TABREAD** block. The column names must be written between quotation marks and separated by a comma. In the **FN 28** block you can define the Q parameter number in which the control is to write the value that is first read.

Use QS parameters if you want to read a text field. Use Q, QL, or QR parameters to read from numerical fields.

Note

If you wish to read from more than one column in an NC block, the control will save the values under successive Q parameters of the same type, such as **QL1**, **QL2**, and **QL3**.

Example

You wish to read the values of the columns **X**, **Y**, and **D** from line 6 of the presently opened table. Save the first value in the Q parameter **Q10**, the second in **Q11**, and the third value in **Q12**.

From the same row, save the column **DOC** in **QS1**.

11 FN 28: TABREAD Q10 = 6/"X,Y,D"	; Read numeric values from columns **X**, **Y** and **D**
12 FN 28: TABREAD QS1 = 6/"DOC"	; Read the alphanumeric value from the **DOC** column

23.2.9 Formulas in the NC program

Application

You can use the **Formula Q/QL/QR** function to define multiple arithmetic operations in one NC block.

Related topics

- String formula for strings

  Further information: "String functions", Page 1301
- Define a calculation in the NC block

  Further information: "Basic arithmetic folder", Page 1278

Description of function

As the first entry, you define the variable to which you assign the result.

To the right of the **=** sign, you define the arithmetic operations.

If you define the functions **Formula Q/QL/QR** or **String Formula QS**, you can open a keyboard for formula input with all available calculation steps in the action bar or form. The virtual keyboard also contains a formula input mode.

Further information: "Virtual keyboard of the control bar", Page 1380

Rules for formulas

Sequence for the evaluation of arithmetic operations

If you enter a mathematical formula that contains more than one mathematical operation, the control always evaluates the individual operations in a defined sequence. A familiar example of this is the rule that multiplication/division takes place before addition/subtraction (higher-level operations are performed first).

The control adheres to the following rules of priority for the evaluation of mathematical formulas:

Priority	Designation	Arithmetic operator
1	Resolve parentheses	**( )**
2	Note the sign, calculate the function	Minus sign, **SIN**, **COS**, **LN** etc.
3	Powers	**^**
4	Multiplication and division	***** , **/**
5	Addition and subtraction	**+**, **-**

Sequence of operations with the same priority

The control generally calculates operations with the same priority from the left to the right.

2 + 3 - 2 = (2 + 3) - 2 = 3

Exception: Concatenated powers are evaluated from right to left.

2 ^ 3 ^ 2 = 2 ^ (3 ^ 2) = 2 ^ 9 = 512

Arithmetic operations

The formula input keyboard includes the following shortcut functions:

Syntax	Linking function	Priority
+	**Addition** e.g. **Q10 = Q1 + Q5**	Addition/subtraction calculation
-	**Subtraction** e.g. **Q25 = Q7 - Q108**	Addition/subtraction calculation
*	**Multiplication** e.g. **Q12 = 5 * Q5**	Multiplication/division calculation
/	**Division** e.g. **Q25 = Q1 / Q2**	Multiplication/division calculation
(	**Opening parenthesis** e.g. **Q12 = Q1 * (Q2 + Q3)**	Expression in parentheses
)	**Closing parenthesis** e.g. **Q12 = Q1 * (Q2 + Q3)**	Expression in parentheses
SQ	**Square of a value** e.g. **Q15 = SQ 5**	Function
SQRT	**Square root** e.g. **Q22 = SQRT 25**	Function
SIN	**Sine of an angle** e.g. **Q44 = SIN 45**	Function
COS	**Cosine of an angle** e.g. **Q45 = COS 45**	Function
TAN	**Tangent of an angle** e.g. **Q46 = TAN 45**	Function
ASIN	**Arc sine** Inverse of the sine. Determines the angle from the ratio of the side opposite the angle and the hypotenuse e.g. **Q10 = ASIN (Q40 / Q20)**	Function
ACOS	**Arc cosine** Inverse of the cosine. Determines the angle from the ratio of the side adjacent to the angle and the hypotenuse e.g. **Q11 = ACOS Q40**	Function
ATAN	**Arc tangent** Inverse of the tangent. Determines the angle from the ratio of the opposite side to the adjacent side e.g. **Q12 = ATAN Q50**	Function
^	**Raising values to a power** e.g. **Q15 = 3 ^ 3**	Power

Syntax	Linking function	Priority
PI	**Pi constant** π = 3.14159 e.g. **Q15 = PI**	
LN	**Natural logarithm (LN) of a number** Base = e = 2.7183 e.g. **Q15 = LN Q11**	Function
LOG	**Logarithm of a number** Base = 10 e.g. **Q33 = LOG Q22**	Function
EXP	**Exponential function (e ^ n)** Base = e = 2.7183 e.g. **Q1 = EXP Q12**	Function
NEG	**Negate values** Multiply by -1 e.g. **Q2 = NEG Q1**	Function
INT	**Truncate decimal places** Form an integer e.g. **Q3 = INT Q42** The **INT** function does not round off—it simply truncates the decimal places.	Function
ABS	**Absolute value of a number** e.g. **Q4 = ABS Q22**	Function
FRAC	**Truncate places before the decimal point** Form a fraction e.g. **Q5 = FRAC Q23**	Function
SGN	**Check algebraic sign of a number** e.g. **Q12 = SGN Q50** If **Q50 = 0**, then **SGN Q50 = 0** If **Q50 < 0**, then **SGN Q50 = -1** If **Q50 > 0**, then **SGN Q50 = 1**	Function
%	**Calculate the modulo value (division remain-der)** e. g., **Q12 = 400 % 360** Result: **Q12 = 40**	Function

You can also define shortcut functions for strings.

Example

Multiplication and division before addition and subtraction

```
11 Q1 = 5 * 3 + 2 * 10 ; Result = 35
```

- 1st calculation: 5 * 3 = 15
- 2nd calculation: 2 * 10 = 20
- 3rd calculation: 15 + 20 = 35

Power before addition and subtraction

11 Q2 = SQ 10 - 3^3	; Result = 73

- 1st calculation: 10 squared = 100
- 2nd calculation: 3 to the power of 3 = 27
- 3rd calculation: 100 – 27 = 73

Function before power

11 Q4 = SIN 30 ^ 2	; Result = 0.25

- 1st calculation: Calculate sine of 30 = 0.5
- 2nd calculation: 0.5 squared = 0.25

Brackets before function

11 Q5 = SIN (50 - 20)	; = 0.5

- 1st calculation: Resolve parentheses: 50 - 20 = 30
- 2nd calculation: Calculate sine of 30 = 0.5

23.3 String functions

Application

With QS parameters, you can define and further process texts, e.g. to create variable logs with **FN 16: F-PRINT**.

Related topics

- Ranges of variables

  Further information: "Variable types", Page 1270

Description of function

You can assign a maximum of 255 characters to a QS parameter.

The following characters are allowed within QS parameters:

- Characters
- Numbers
- Special characters, for example **?**
- Control characters, for example **** for paths
- Spaces

The individual string functions are programmed using the free syntax input.

Further information: "Editing NC functions", Page 205

You can further process or check the values of QS parameters with the functions **Formula Q/QL/QR** and **String Formula QS**.

Syntax	Function	Higher-level NC function
DECLARE STRING	Assigning a value to QS parameter **Further information:** "Assigning text to a QS parameter", Page 1305	
STRING FORMULA	Linking QS parameters **Further information:** "Linking QS parameters", Page 1305	**String formula QS**

Syntax	Function	Higher-level NC function
TONUMB	Converting the alphanumeric value of a QS parameter to a numeric value and assign it to a variable **Further information:** "Converting variable text content to numeric values ", Page 1306	**Formula Q/QL/QR**
TOCHAR	Converting the numeric value to an alphanumeric value and assign it to a QS parameter **Further information:** "Converting variable numeric values to text content", Page 1306	**String formula QS**
SUBSTR	Copying a substring from a QS parameter **Further information:** "Copying a substring from a QS parameter", Page 1306	**String formula QS**
SYSSTR	Read system data **Further information:** "Read system data with SYSSTR", Page 1303	**String formula QS**
INSTR	Checking if the content of one QS parameter is contained in another QS parameter. **Further information:** "Searching for a substring within the content of a QS parameter", Page 1306	**Formula Q/QL/QR**
STRLEN	Determining the text length of the value of a QS parameter **Further information:** "Determining the total number of characters in a QS parameter", Page 1306 If the selected string parameter is not defined the control returns the result **-1**.	**Formula Q/QL/QR**
STRCOMP	Compare alphabetic priority **Further information:** "Comparing the alphabetical order of two QS parameter contents", Page 1307	**Formula Q/QL/QR**
CFGREAD	Read out machine parameter **Further information:** "Accepting the contents of a machine parameter", Page 1307	■ **String formula QS** ■ **Formula Q/QL/QR**

Read system data with SYSSTR

With the function **SYSSTR** you can read system data and store them in string parameters. You select the system data through a group number (ID) and a number.

Entering IDX and DAT is not required.

You can read the following system data:

Group name, ID no.	Number	Meaning
Program information, 10010	1	Path of the current main program or pallet program
	2	Path of the NC program shown in the block display
	3	Path of the cycle selected with **CYCL DEF 12 PGM CALL**
	10	Path of the NC program selected with **SEL PGM**
Channel data, 10025	1	Channel name
Values programmed in the tool call, 10060	1	Tool name
Kinematics, 10290	10	Kinematics programmed in the last **FUNCTION MODE** block
Current system time, 10321	1 to 16, 20	■ 1: DD.MM.YYYY hh:mm:ss ■ 2 and 16: DD.MM.YYYY hh:mm ■ 3: DD.MM.YY hh:mm ■ 4: YYYY-MM-DD hh:mm:ss ■ 5 and 6: YYYY-MM-DD hh:mm ■ 7: YY-MM-DD hh:mm ■ 8 and 9: DD.MM.YYYY ■ 10: D.MM.YY ■ 11: YYYY-MM-DD ■ 12: YY-MM-DD ■ 13 and 14: hh:mm:ss ■ 15: hh:mm ■ 20: XX "XX" stands for the two-digit number of the current calendar week that—in accordance with ISO 8601 —is characterized by the following: ■ It comprises seven days ■ It begins with Monday ■ It is numbered sequentially ■ The first calendar week (week 01) is the week with the first Thursday of the Gregorian year.
Touch-probe data, 10350	50	Probe type of the active touch probe TS
	70	Probe type of the active touch probe TT
	73	Key name of the active touch probe TT from MP **activeTT**

Group name, ID no.	Number	Meaning
Data for pallet machining, 10510	1	Name of the pallet being machined
	2	Path of the selected pallet table
NC software version, 10630	10	Version identifier of the NC software version
Information for unbalance cycle, 10855	1	Path of the unbalance calibration table belonging to the active kinematics
Tool data, 10950	1	Tool name
	2	DOC entry of the tool
	3	AFC control setting
	4	Tool-carrier kinematics

Read machine parameters with CFGREAD

With the **CFGREAD** function, you can read out machine parameters of the control as numerical values or as strings. The read-out values are always output in metric units of measure.

To read a machine parameter, you must determine the following contents in the configuration editor:

- Parameter name
- Parameter object
- If present, group name and index

Further information: "Accepting the contents of a machine parameter", Page 1307

Symbol	Type	Meaning	Example
	Key	Group name of the machine parameter (if available)	**CH_NC**
	Entity	Parameter object (name begins with **Cfg**...)	**CfgGeoCycle**
	Attribute	Name of the machine parameter	**displaySpindleErr**
	Index	List index of a machine parameter (if available)	**[0]**

If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts.

Each time you want to interrogate a machine parameter with the **CFGREAD** function, you must first define a QS parameter with attribute, entity and key.

The following parameters are read in the CFGREAD function's dialog:

- **KEY_QS**: Group name (key) of the machine parameter
- **TAG_QS**: Object name (entity) of the machine parameter
- **ATR_QS**: Name (attribute) of the machine parameter
- **IDX**: Index of the machine parameter

23.3.1 Assigning text to a QS parameter

Before you can use and further process texts, you must assign the characters to QS parameters. Use the **DECLARE STRING** command to do so.

To assign a text to a QS parameter:

Insert NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **DECLARE STRING**
- ▶ Define QS parameters for the result, e.g. **QS10**
- ▶ Select **Name**
- ▶ Enter the desired text
- ▶ End NC block
- ▶ Execute NC block
- > The control assigns the entered text to the QS parameter.

In this example, a text is assigned to the QS parameter **QS10**.

```
37 DECLARE STRING QS10 = "workpiece"
```

23.3.2 Linking QS parameters

You can use the concatenation operator **||** to link the characters of multiple QS parameters. This allows you to combine fixed and variable text elements.

To connect the values of multiple QS parameters:

Insert NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **String formula QS**
- ▶ Define QS parameters for the result
- ▶ Open keyboard for formula input
- ▶ Select concatenation operator **||**
- ▶ To the left of the concatenation operator icon, define the number of the QS parameter with the first substring
- ▶ To the right of the concatenation operator icon, define the number of the QS parameter with the second substring
- ▶ End NC block
- ▶ Confirm your input
- > After execution, the control saves the substrings in succession as a value in the target parameter.

In this example, **QS10** should contain the complete text of **QS12**, **QS13**, and **QS14**.

```
37 QS10 =  QS12 || QS13 || QS14
```

Parameter contents:

- **QS12**: **Workpiece**
- **QS13**: **Status:**
- **QS14**: **Scrap**
- **QS10**: **Workpiece Status: Scrap**

23.3.3 Converting variable text content to numeric values

The **TONUMB** function allows you to store numeric characters of a QS parameter as the value of a variable. The value to be converted can only consist of numbers. The stored value can be used, for example, to perform calculations.

In this example, the QS parameter **QS11** is converted to the numerical parameter **Q82**.

```
37 Q82 = TONUMB ( SRC_QS11 )
```

23.3.4 Converting variable numeric values to text content

You can use the **TOCHAR** function to save the contents of a variable to a QS parameter. You can link the saved content with other QS parameters, for example.

In this example, the content of the numeric parameter **Q50** is transferred to the string parameter **QS11**.

```
37 QS11 = TOCHAR ( DAT+Q50 DECIMALS3 )
```

23.3.5 Copying a substring from a QS parameter

The **SUBSTR** function allows you to save a definable range from one QS parameter to another QS parameter. For example, you can use this function to extract the file name from an absolute file path.

In this example, the syntax element **BEG2** is used to read a four-character substring from the third character, since counting starts from zero and **LEN4** is commanded.

```
37 QS13 = SUBSTR ( SRC_QS10 BEG2 LEN4 )
```

23.3.6 Searching for a substring within the content of a QS parameter

You can use the **INSTR** function to check whether a specific substring is within a QS parameter. This allows you to check, for example, whether the concatenation of multiple QS parameters has worked. Two QS parameters are required for the check. The control searches the first QS parameter for the content of the second QS parameter.

If the control cannot find the substring to be searched for, the control saves the total number of characters to the result parameter.

If the substring to be searched for appears multiple times, then the control returns the first place at which it finds the substring.

In this example, **QS10** is searched for in the text stored in **QS13**. The search starts from the third character. When counting the characters, start with zero.

```
37 Q50 = INSTR ( SRC_QS10 SEA_QS13 BEG2 )
```

23.3.7 Determining the total number of characters in a QS parameter

The **STRLEN** function returns the length of the text stored in a selectable QS parameter. You can use this function to determine the length of a file path, for example.

In this example, the length of **QS15** is determined.

```
37 Q52 = STRLEN ( SRC_QS15 )
```

23.3.8 Comparing the alphabetical order of two QS parameter contents

The **STRCOMP** function allows you to compare the alphabetical order of the contents of two QS parameters. For example, you can use this function to check whether a QS parameter contains uppercase or lowercase letters. The control first searches all uppercase letters alphabetically and then all lowercase letters alphabetically.

The control returns the following results:

- **0**: The compared QS parameters are identical
- **-1**: The first QS parameter **precedes** the second QS parameter alphabetically
- **+1:** The first QS parameter **follows** the second QS parameter alphabetically

This example compares the alphabetical order of **QS12** and **QS14**.

```
37 Q52 = STRCOMP  ( SRC_QS12 SEA_QS14 )
```

23.3.9 Accepting the contents of a machine parameter

You can use the NC function **CFGREAD** to transfer the content of a machine parameter to a QS parameter.

Depending on the content of the machine parameter, you can use the **CFGREAD** function to copy text content to QS parameters or numeric values to Q, QL or QR parameters.

In this example, the axis designation of the fourth axis is loaded as a QS parameter.

Specified settings in the machine parameters:

- DisplaySettings
- CfgDisplayData
  - axisDisplayOrder
  - [0] to [5]

Example

14 QS11 = ""	; Assign QS parameter for the key
15 QS12 = "CfgDisplaydata"	; Assign QS parameter for the entity
16 QS13 = "axisDisplay"	; Assign QS parameter for the parameter name
17 QS1 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13 IDX3)	; Read out machine parameter(s)

Note

When you use the **STRING FORMULA** function, the result of the performed arithmetic operation is always a string. When you use the **FORMULA** function, the result of the performed arithmetic operation is always a numeric value.

Definition

String
In computer science, a string is a defined sequence of alphanumeric characters, i.e. texts. The control uses QS parameters to process strings.

23.4 Defining counters with FUNCTION COUNT

Application

The **FUNCTION COUNT** function allows you to control a simple counter from within the NC program. For example, this function allows you to count the number of manufactured workpieces.

Description of function

The count is retained even after a restart of the control.

The control only takes the **FUNCTION COUNT** function into account in the **Program Run** operating mode.

The control shows the current counter value and the defined target number on the **PGM** tab of the **Status** workspace.

Further information: "PGM tab", Page 156

Input

```
11 FUNCTION COUNT TARGET5    ; Set the target value of the counter to 5
```

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION COUNT	Syntax initiator for the counter
INC, **RESET**, **ADD**, **SET**, **TARGET** or **REPEAT**	Define counting function **Further information:** "Counting functions", Page 1308

Counting functions

The **FUNCTION COUNT** function provides the following possibilities:

Syntax	Function
INC	Increase count by 1
RESET	Reset counter
ADD	Increment the counter by the desired value Input: **0...9999**
SET	Set the counter to the desired value Input: **0...9999**
TARGET	Set the nominal count (target value) to the desired value Input: **0...9999**
REPEAT	Repeat the NC program from the defined label if the target value has not yet been reached. Fixed or variable number or name

Notes

NOTICE

Caution: Data may be lost!

Only one counter can be managed by the control. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted.

- Please check prior to machining whether a counter is active.

- The machine manufacturer uses the optional machine parameter **CfgNcCounter** (no. 129100) to define whether you can edit the counter.
- You can use Cycle **225** to engrave the current counter value into the workpiece.

  Further information: "Cycle 225 ENGRAVING ", Page 638

23.4.1 Example

11 FUNCTION COUNT RESET	; Reset counter value
12 FUNCTION COUNT TARGET10	; Set target number of machining operations
13 LBL 11	; Jump label
*** - ...**	; Machining operation
21 FUNCTION COUNT INC	; Increase counter value
22 FUNCTION COUNT REPEAT LBL 11	; Repeat machining operation if the target number has not been reached
23 M30	
24 END PGM	

23.5 Program defaults for cycles

23.5.1 Overview

Some cycles always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. With the **GLOBAL DEF** function you can define these cycle parameters at the beginning of the program, so that they are effective globally for all cycles used in the NC program. In the respective cycle you then use **PREDEF** to simply reference the value defined at the beginning of the program.

The following **GLOBAL DEF** functions are available

Cycle		Activation	Further information
100	**GENERAL** Definition of generally valid cycle parameters ■ **Q200 SET-UP CLEARANCE** ■ **Q204 2ND SET-UP CLEARANCE** ■ **Q253 F PRE-POSITIONING** ■ **Q208 RETRACTION FEED RATE**	**DEF**-active	Page 1312
105	**DRILLING** Definition of specific drilling cycle parameters ■ **Q256 DIST FOR CHIP BRKNG** ■ **Q210 DWELL TIME AT TOP** ■ **Q211 DWELL TIME AT DEPTH**	**DEF**-active	Page 1313
110	**POCKET MILLING** Definition of specific pocket-milling cycle parameters ■ **Q370 TOOL PATH OVERLAP** ■ **Q351 CLIMB OR UP-CUT** ■ **Q366 PLUNGE**	**DEF**-active	Page 1314
111	**CONTOUR MILLING** Definition of specific contour-milling cycle parameters ■ **Q2 TOOL PATH OVERLAP** ■ **Q6 SET-UP CLEARANCE** ■ **Q7 CLEARANCE HEIGHT** ■ **Q9 ROTATIONAL DIRECTION**	**DEF**-active	Page 1315
125	**POSITIONING** Definition of the positioning behavior with **CYCL CALL PAT** ■ **Q345 SELECT POS. HEIGHT**	**DEF**-active	Page 1315
120	**PROBING** Definition of specific touch probe cycle parameters ■ **Q320 SET-UP CLEARANCE** ■ **Q260 CLEARANCE HEIGHT** ■ **Q301 MOVE TO CLEARANCE**	**DEF**-active	Page 1316

23.5.2 Entering GLOBAL DEF definitions

Insert NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select **GLOBAL DEF**
- ▶ Select the desired **GLOBAL DEF** function, e.g. **100 GENERAL**
- ▶ Enter the required definitions

23.5.3 Using GLOBAL DEF information

If you entered the corresponding **GLOBAL DEF** functions at program start, you can reference these globally valid values for the definition of any cycle.

Proceed as follows:

Insert NC function

- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- ▶ Select and define **GLOBAL DEF**
- ▶ Select **Insert NC function** again
- ▶ Select the desired cycle, e.g. **200 DRILLING**
- > If the cycle includes global cycle parameters, the control superimposes the selection possibility **PREDEF** in the action bar or in the form as a selection menu.

PREDEF

- ▶ Select **PREDEF**
- > The control then enters the word **PREDEF** in the cycle definition. This creates a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program.

NOTICE

Danger of collision!

If you later edit the program settings with **GLOBAL DEF**, these changes will affect the entire NC program. This may change the machining sequence significantly.

- ▶ Make sure to use **GLOBAL DEF** carefully. Simulate your program before executing it
- ▶ If you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

23.5.4 Global data valid everywhere

The parameters are valid for all **2xx** machining cycles as well as for Cycles **880,**
1017, **1018**, **1021**, **1022**, **1025** and touch probe cycles **451**, **452**, **453**

Help graphic	Parameter
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between touch probe and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Feed rate at which the control moves the tool within a cycle. Input: **0...99999.999** or **FMAX**, **FAUTO**
	Q208 Feed rate for retraction? Feed rate at which the control retracts the tool. Input: **0...99999.999** or **FMAX**, **FAUTO**

Example

```
11 GLOBAL DEF 100 GENERAL ~
    Q200=+2                  ;SET-UP CLEARANCE ~
    Q204=+50                 ;2ND SET-UP CLEARANCE ~
    Q253=+750                ;F PRE-POSITIONING ~
    Q208=+999                ;RETRACTION FEED RATE
```

23.5.5 Global data for drilling operations

The parameters apply to the drilling, tapping, and thread milling cycles **200** to **209**, **240**, **241**, **262** to **267**.

Help graphic	Parameter
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: **0.1...99999.9999**
	Q210 Dwell time at the top? Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input: **0...3600.0000**
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: **0...3600.0000**

Example

```
11 GLOBAL DEF 105 DRILLING ~
    Q256=+0.2               ;DIST FOR CHIP BRKNG ~
    Q210=+0                 ;DWELL TIME AT TOP ~
    Q211=+0                 ;DWELL TIME AT DEPTH
```

23.5.6 Global data for milling operations with pocket cycles

The parameters apply to the cycles **208**, **232**, **233**, **251** to **258**, **262** to **264**, **267**, **272**, **273**, **275**, and **277**

Help graphic	Parameter
	Q370 Path overlap factor? **Q370** x tool radius = stepover factor k. Input: **0.1...1999**
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. **+1** = climb milling **–1** = up-cut milling (If you enter 0, climb milling is performed.) Input: **-1**, **0**, **+1**
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: **0**: Vertical plunging. The control plunges perpendicularly, regardless of the plunging angle **ANGLE** defined in the tool table. **1**: Helical plunging. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message **2**: Reciprocating plunge. In the tool table, the plunging angle **ANGLE** for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. The reciprocation length depends on the plunging angle. As a minimum value the control uses twice the tool diameter. Input: **0**, **1**, **2**

Example

```
11 GLOBAL DEF 110 POCKET MILLING ~
    Q370=+1                    ;TOOL PATH OVERLAP ~
    Q351=+1                    ;CLIMB OR UP-CUT ~
    Q366=+1                    ;PLUNGE
```

23.5.7 Global data for milling operations with contour cycles

The parameters apply to the cycles **20**, **24**, **25**, **27** to **29**, **39**, and **276**

Help graphic	Parameter
	Q2 Path overlap factor? **Q2** x tool radius = stepover factor k Input: **0.0001...1.9999**
	Q6 Set-up clearance? Distance between tool tip and the top surface of the workpiece. This value has an incremental effect. Input: **-99999.9999...+99999.9999**
	Q7 Clearance height? Height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle). The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q9 Direction of rotation? cw = -1 Machining direction for pockets ■ **Q9** = –1 up-cut milling for pocket and island ■ **Q9** = +1 climb milling for pocket and island Input: **-1**, **0**, **+1**

Example

```
11 GLOBAL DEF 111 CONTOUR MILLING ~
    Q2=+1                   ;TOOL PATH OVERLAP ~
    Q6=+2                   ;SET-UP CLEARANCE ~
    Q7=+50                  ;CLEARANCE HEIGHT ~
    Q9=+1                   ;ROTATIONAL DIRECTION
```

23.5.8 Global data for positioning behavior

The parameters apply to each fixed cycle that you call with the **CYCL CALL PAT** function.

Help graphic	Parameter
	Q345 Select positioning height (0/1) Retraction in the tool axis at the end of a machining step, return to the 2nd set-up clearance or to the position at the beginning of the unit. Input: **0**, **1**

Example

```
11 GLOBAL DEF 125 POSITIONING ~
    Q345=+1                 ;SELECT POS. HEIGHT
```

23.5.9 Global data for probing functions

The parameters are valid for all touch probe cycles **4xx** and **14xx** as well as for Cycles **271**, **286**, **287**, **880**, **1021**, **1022**, **1025**, **1271**, **1272**, **1273**, **1278**

Help graphic	Parameter
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**

Example

```
11 GLOBAL DEF 120 PROBING ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q301=+1                 ;MOVE TO CLEARANCE
```

23.6 Table access with SQL statements

23.6.1 Fundamentals

Application

If you would like to access numerical or alphanumerical content in a table or manipulate the table (e.g., rename columns or rows), then use the available SQL commands.

The syntax of the SQL commands available on the control is strongly influenced by the SQL programming language but does not conform with it entirely. In addition, the control does not support the full scope of the SQL language.

Related topics

- Opening, reading and writing to freely definable tables

  Further information: "Functions for freely definable tables", Page 1296

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function

In the NC software, table accesses occur through an SQL server. This server is controlled via the available SQL commands. The SQL commands can be defined directly in an NC program.

The server is based on a transaction model. A **transaction** consists of multiple steps that are executed together, thereby ensuring that the table entries are processed in an orderly and well-defined manner.

The SQL commands take effect in the **Program Run** operating mode and the **MDI** application.

Example of transaction:

- Assign Q parameters to table columns for read or write access using **SQL BIND**
- Select data using **SQL EXECUTE** with the instruction **SELECT**
- Read, change, or add data using **SQL FETCH**, **SQL UPDATE**, or **SQL INSERT**
- Confirm or discard interaction using **SQL COMMIT** or **SQL ROLLBACK**
- Approve bindings between table columns and Q parameters using **SQL BIND**

> You must conclude all transactions that have been started—even exclusively reading accesses. Concluding the transaction is the only way to ensure that changes and additions are transferred, that locks are removed, and that used resources are released.

The **result set** contains a subset of a table file. It results from a **SELECT** query performed on the table.

The **result set** is created when a query is executed in the SQL server, thereby occupying resources there.

This query has the same effect as applying a filter to the table, so that only part of the data records become visible. To perform this query, the table file must be read at this point.

The SQL server assigns a **handle** to the **result set**, which enables you to identify the result set for reading or editing data and completing the transaction. The **handle** is the result of the query, which is visible in the NC program. The value 0 indicates an **invalid handle**, i.e. it was not possible to create a **result set** for that query. If no rows are found that satisfy the specified condition, an empty **result set** is created and assigned a valid **handle.**

Overview of SQL commands

The control provides the following SQL commands:

Syntax	Function	Further information
SQL BIND	**SQL BIND** creates or disconnects a binding between table columns and Q or QS parameters	Page 1318
SQL SELECT	**SQL SELECT** reads out a single value from a table and does not open any transaction	Page 1320
SQL EXECUTE	**SQL EXECUTE** opens a transaction for selected table columns and table rows or enables the use of other SQL instructions (miscellaneous functions).	Page 1322
SQL FETCH	**SQL FETCH** transfers the values to the bound Q parameters	Page 1326
SQL ROLLBACK	**SQL ROLLBACK** discards all changes and concludes the transaction	Page 1327
SQL COMMIT	**SQL COMMIT** saves all changes and concludes the transaction	Page 1329
SQL UPDATE	**SQL UPDATE** expands the transaction to include the change of an existing row	Page 1330
SQL INSERT	**SQL INSERT** creates a new table row	Page 1332

Notes

NOTICE

Danger of collision!

Read and write accesses performed with the help of SQL commands always occur in metric units, regardless of the unit of measure selected for the table or the NC program.
If, for example, you save a length from a table to a Q parameter, then the value is thereafter always in metric units. If this value is then used for the purpose of positioning in an inch program (**L X+Q1800**), then an incorrect position will result.

- In inch programs, convert the read value prior to use

- HEIDENHAIN recommends that you use SQL functions instead of **FN 26**, **FN 27**, or **FN 28** in order to achieve maximum HDR hard-disk speeds for table applications and to reduce the amount of computing power used.

23.6.2 Binding a variable to a table column with SQL BIND

Application

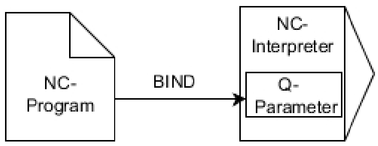
SQL BIND links a Q parameter to a table column. The SQL commands **FETCH**, **UPDATE**, and **INSERT** evaluate this binding (assignment) during data transfer between the **result set** and the NC program.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function



Program any number of bindings with **SQL BIND...**, before using the **FETCH**, **UPDATE**, or **INSERT** commands.

An **SQL BIND** command without a table name or column name cancels the binding. At the latest, the binding is terminated at the end of the NC program or subprogram.

Input

11 SQL BIND Q881 "Tab_example.Position_Nr"	; Bind **Q881** to the "Position_No" column of the "Tab_Example" table

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL BIND	Syntax initiator for the **BIND** SQL command
Q/QL/QR, **QS** or **Q REF**	Variable to be bound
" " or **QS**	Table name and table column, separated by **.** or QS parameter with definition

Notes

- Enter the path of the table or a synonym as the table name.

  Further information: "Executing SQL statements with SQL EXECUTE", Page 1322
- During the read and write operations, the control considers only those columns that you have specified by means of the **SELECT** command. If you specify columns without a binding in the **SELECT** command, then the control interrupts the read or write operation with an error message.

23.6.3 Reading out a table value with SQL SELECT

Application

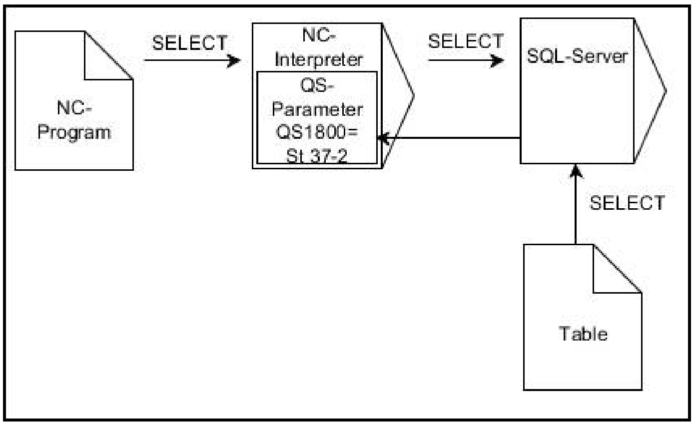
SQL SELECT reads a single value from a table and saves the result in the defined Q parameter.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function



Black arrows and associated syntax show internal processes of **SQL SELECT**

With **SQL SELECT**, there is neither a transaction nor a binding between the table column and Q parameter. The control does not consider any bindings that may exist to the specified column. The control copies the read value only into the parameter specified for the result.

Input

11 SQL SELECT Q5 "SELECT Mess_X FROM Tab_Example WHERE Position_NR==3"	; Save the value of the "Position_No" column of the "Tab_Example" table in **Q5**

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL BIND	Syntax initiator for the **SELECT** SQL command
Q/QL/QR, **QS** or **Q REF**	Variable in which the control stores the result
" " or **QS**	SQL statement or QS parameter with the definition containing: ■ **SELECT**: Table column of the value to be transferred ■ **FROM**: Synonym or absolute path of the table (path in single quotation marks) ■ **WHERE**: Column designation, condition, and comparison value (Q parameter after **:** in single quotation marks)

Notes

- You can select multiple values or multiple columns using the SQL command **SQL EXECUTE** and the **SELECT** statement.
- For the instructions within the SQL command, you can likewise use single or combined QS parameters.

  Further information: "Linking QS parameters", Page 1305
- If you check the content of a QS parameter in the additional status indicator (**QPARA** tab), then you will see only the first 30 characters and therefore not the entire content.

  Further information: "QPARA tab", Page 158

Example

The result of the following NC programs is identical.

0 BEGIN PGM SQL_READ_WMAT MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table \WMAT.TAB'"	; Create synonym
2 SQL BIND QS1800 "my_table.WMAT"	; Bind QS parameters
3 SQL QL1 "SELECT WMAT FROM my_table WHERE NR==3"	; Define search
*** - ...**	

*** - ...**	
3 SQL SELECT QS1800 "SELECT WMAT FROM my_table WHERE NR==3"	; Read and save value
*** - ...**	

*** - ...**	
3 DECLARE STRING QS1 = "SELECT "	
4 DECLARE STRING QS2 = "WMAT "	
5 DECLARE STRING QS3 = "FROM "	
6 DECLARE STRING QS4 = "my_table "	
7 DECLARE STRING QS5 = "WHERE "	
8 DECLARE STRING QS6 = "NR==3"	
9 QS7 = QS1 || QS2 || QS3 || QS4 || QS5 || QS6	
10 SQL SELECT QL1 QS7	
*** - ...**	

23.6.4 Executing SQL statements with SQL EXECUTE

Application

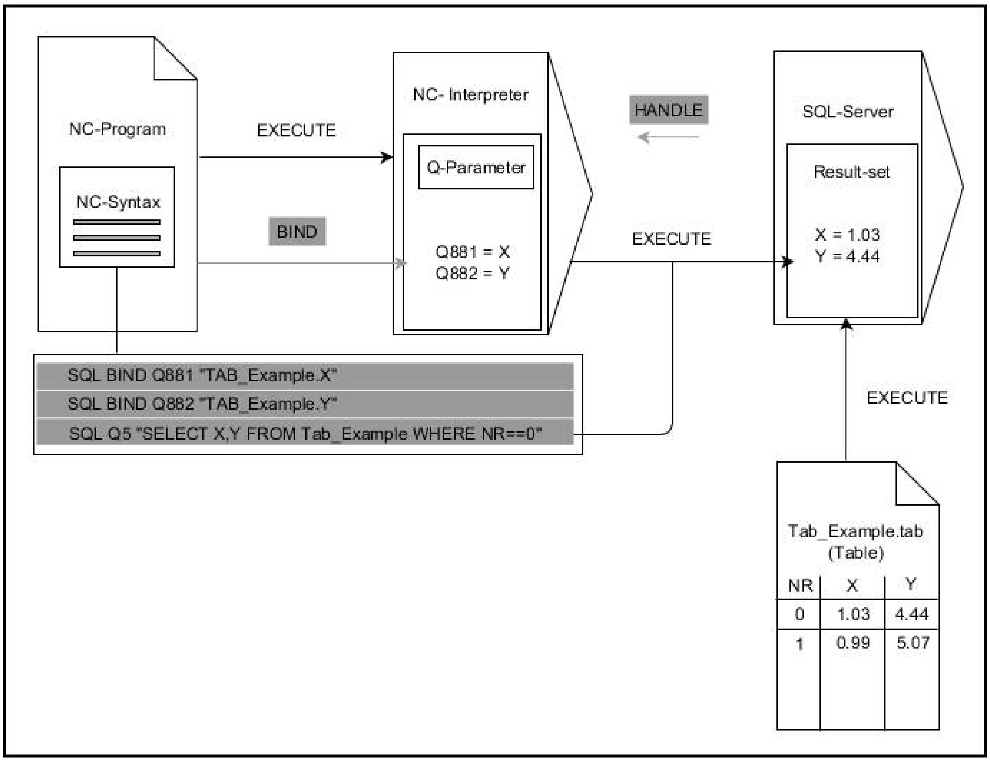
SQL EXECUTE can be used in conjunction with various SQL instructions.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function



Black arrows and associated syntax indicate internal processes of **SQL EXECUTE**. The gray arrows and associated syntax do not directly belong to the **SQL EXECUTE** command.

The control provides the following SQL statements in the **SQL EXECUTE** command:

Instruction	Function
SELECT	Select data
CREATE SYNONYM	Create synonym (replace long path names with short names)
DROP SYNONYM	Delete synonym
CREATE TABLE	Generate a table
COPY TABLE	Copy table
RENAME TABLE	Rename table
DROP TABLE	Delete a table
INSERT	Insert table rows
UPDATE	Update table rows
DELETE	Delete table rows
ALTER TABLE	■ Add table columns using **ADD** ■ Delete table columns using **DROP**
RENAME COLUMN	Rename table columns

SQL EXECUTE with the SQL instruction SELECT

The SQL server places the data in the **result set** row-by-row. The rows are numbered in ascending order, starting with 0. The SQL commands **FETCH** and **UPDATE** use these row numbers (the **INDEX**).

SQL EXECUTE, in conjunction with the SQL instruction **SELECT**, selects the table values, transfers them to the **result set**, and always opens a transaction in the process. Unlike the SQL command **SQL SELECT**, the combination of **SQL EXECUTE** and the **SELECT** instruction allows multiple columns and rows to be selected at the same time.

In the function **SQL ... “SELECT...WHERE..."**, you can enter the search criteria. You thereby restrict the number of rows to be transferred. If you do not use this option, then all of the rows in the table are loaded.

In the function **SQL ... “SELECT...ORDER BY..."**, you can enter the ordering criterion. This entry consists of the column designation and the keyword **ASC** for ascending or **DESC** for descending order. If you do not use this option, then rows will be stored in a random order.

With the function **SQL ... "SELECT...FOR UPDATE"**, you can lock the selected rows for other applications. Other applications can continue to read these rows but are unable to change them. If you make changes to the table entries, then it is absolutely necessary to use this option.

Empty result set: If no rows meet the search criterion, then the SQL server returns a valid **HANDLE** without table entries.

Conditions for WHERE entries

Condition	Programming
Equals	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
Empty	IS NULL
Not empty	IS NOT NULL
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

Notes

- You can also define synonyms for tables that have not yet been generated.
- The sequence of the columns in the created file corresponds to the sequence within the **AS SELECT** instruction.
- For the instructions within the SQL command, you can likewise use single or combined QS parameters.

  Further information: "Linking QS parameters", Page 1305
- If you check the content of a QS parameter in the additional status indicator (**QPARA** tab), then you will see only the first 30 characters and therefore not the entire content.

  Further information: "QPARA tab", Page 158

Example

Example: selection of table rows

11 SQL BIND Q881 "Tab_Example.Position_Nr"	
12 SQL BIND Q882 "Tab_Example.Measure_X"	
13 SQL BIND Q883 "Tab_Example.Measure_Y"	
14 SQL BIND Q884 "Tab_Example.Measure_Z"	
. . .	
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example"	

Example: Select table rows with the WHERE function

20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example WHERE Position_Nr<20"	

Example: Select table rows with the WHERE function and Q parameter

20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example WHERE Position_Nr==:'Q11'"	

Example: Define the table name with absolute path information

20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM 'V:\table\Tab_Example' WHERE Position_Nr<20"	

0 BEGIN PGM SQL_CREATE_TAB MM	
1 SQL Q10 "CREATE SYNONYM NEW FOR 'TNC:\table\NewTab.TAB'"	Create synonym
2 SQL Q10 "CREATE TABLE NEW AS SELECT X,Y,Z FROM 'TNC:\prototype_for_NewTab.tab'"	Create table
3 END PGM SQL_CREATE_TAB MM	

0 BEGIN PGM SQL_CREATE_TABLE_QS MM	
1 DECLARE STRING QS1 = "CREATE TABLE "	
2 DECLARE STRING QS2 = "'TNC:\nc_prog\demo\Doku\NewTab.t' "	
3 DECLARE STRING QS3 = "AS SELECT "	
4 DECLARE STRING QS4 = "DL,R,DR,L "	
5 DECLARE STRING QS5 = "FROM "	
6 DECLARE STRING QS6 = "'TNC:\table\tool.t'"	
7 QS7 = QS1 || QS2 || QS3 || QS4 || QS5 || QS6	
8 SQL Q1800 QS7	
9 END PGM SQL_CREATE_TABLE_QS MM	

23.6.5 Reading a line from a result set with SQL FETCH

Application

SQL FETCH reads a row from the **result set**. The values of the individual cells are stored by the control in the bound Q parameters. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**.

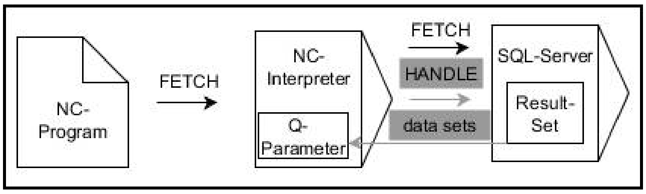
SQL FETCH takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function



Black arrows and associated syntax indicate internal processes of **SQL FETCH**. The gray arrows and associated syntax do not directly belong to the **SQL FETCH** command.

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

11 SQL FETCH Q1 HANDLE Q5 INDEX 5 IGNORE UNBOUND UNDEFINE MISSING	; Read out result of transaction **Q5** line 5

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL FETCH	Syntax initiator for the **FETCH** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of the transaction
INDEX	Row number within the **result set** as a number or variable If not specified, the control accesses line 0. Optional syntax element
IGNORE UNBOUND	For the machine manufacturer only Optional syntax element
UNDEFINE MISSING	For the machine manufacturer only Optional syntax element

Example

Transfer line number in the Q parameter

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM
   Tab_Example"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

23.6.6 Discarding changes to a transaction using SQL ROLLBACK

Application

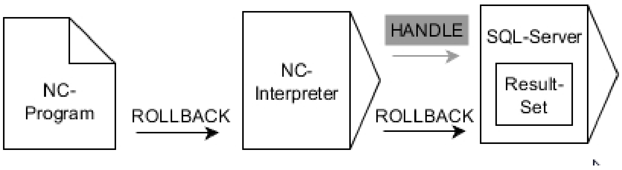
SQL ROLLBACK discards all of the changes and additions of a transaction. The transaction is defined via the **HANDLE** to be specified.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function



Black arrows and associated syntax indicate internal processes of **SQL ROLLBACK**. The gray arrows and associated syntax do not directly belong to the **SQL ROLLBACK** command.

The function of the SQL command **SQL ROLLBACK** depends on the **INDEX**:

- Without **INDEX**:
  - The control discards all changes and additions of the transaction
  - The control resets a lock set with **SELECT...FOR UPDATE**
  - The control completes the transaction (the **HANDLE** loses its validity)
- With **INDEX**:
  - Only the indexed row remains in the **result set** (the control removes all of the other rows)
  - The control discards any changes and additions that may have been made in the non-specified rows
  - The control locks only those rows indexed with **SELECT...FOR UPDATE** (the control resets all of the other locks)
  - The specified (indexed) row is then the new Row 0 of the **result set**
  - The control does **not** complete the transaction (the **HANDLE** keeps its validity)
  - The transaction must be completed manually with **SQL ROLLBACK** or **SQL COMMIT** at a later time

Input

11 SQL ROLLBACK Q1 HANDLE Q5 INDEX 5	; Delete all rows of transaction **Q5** except row 5

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL ROLLBACK	Syntax initiator for the **ROLLBACK** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of the transaction
INDEX	Row number within the **Result set** as a number or variable that is retained If not specified, the control discards all changes and additions to the transaction Optional syntax element

Example

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM
   Tab_Example"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
* - ...
41 SQL ROLLBACK Q1 HANDLE Q5
```

23.6.7 Completing a transaction with SQL COMMIT

Application

SQL COMMIT simultaneously transfers all of the rows that have been changed and added in a transaction back into the table. The transaction is defined via the **HANDLE** to be specified. In this context, a lock that has been set with **SELECT...FOR UPDATE** resets the control.

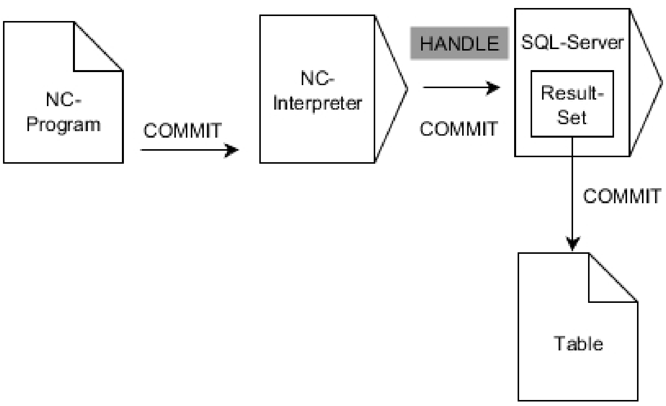
Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function

The assigned **HANDLE** (operation) loses its validity.



Black arrows and associated syntax indicate internal processes of **SQL COMMIT**.

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

11 SQL COMMIT Q1 HANDLE Q5	; Complete all rows of transaction **Q5** and update table

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL COMMIT	Syntax initiator for the **COMMIT** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of the transaction

Example

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM
    Tab_Example"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
* - ...
41 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
* - ...
51 SQL COMMIT Q1 HANDLE Q5
```

23.6.8 Changing the row of a result set with SQL UPDATE

Application

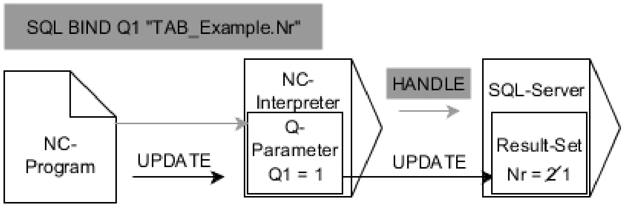
SQL UPDATE changes a row in the **result set**. The new values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**. The control completely overwrites the already existing rows in the **result set**.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function



Black arrows and the associated syntax show internal **SQL UPDATE** processes. Gray arrows and the associated syntax are not directly associated with the **SQL UPDATE** command.

SQL UPDATE takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

11 SQL UPDATE Q1 HANDLE Q5 index5 RESET UNBOUND	; Complete all rows of transaction **Q5** and update table

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL UPDATE	Syntax initiator for the **UPDATE** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of the transaction
INDEX	Row number within the **Result set** as a number or variable If not specified, the control accesses line 0. Optional syntax element
RESET UNBOUND	For the machine manufacturer only Optional syntax element

Note

When writing to tables, the control checks the lengths of the string parameters. If the entries exceed the length of the columns to be described, then the control outputs an error message.

Example

Transfer line number in the Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.Position_NR"
12 SQL BIND Q882 "TAB_EXAMPLE.Measure_X"
13 SQL BIND Q883 "TAB_EXAMPLE.Measure_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.Measure_Z"
* - ...
21 SQL Q5 "SELECT Position_NR,Measure_X,Measure_Y,Measure_Z FROM
   TAB_EXAMPLE"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Program the row number directly

```
31 SQL UPDATE Q1 HANDLE Q5 INDEX5
```

23.6.9 Creating a new row in the result set with SQL INSERT

Application

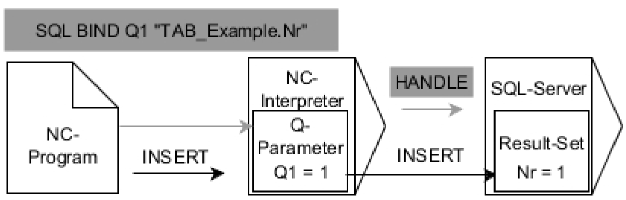
SQL INSERT creates a new row in the **result set**. The values of the individual cells are copied by the control from the bound Q parameters. The transaction is defined via the **HANDLE** to be specified.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

  The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Description of function



Black arrows and associated syntax indicate internal processes of **SQL INSERT**. The gray arrows and associated syntax do not directly belong to the **SQL INSERT** command.

SQL INSERT takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**). Table columns without a corresponding **SELECT** instruction (not contained in the query result) are described by the control with default values.

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

11 SQL INSERT Q1 HANDLE Q5	; Create a new row in transaction **Q5**

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL INSERT	Syntax initiator for the **INSERT** SQL command
Q/QL/QR or **Q REF**	Variable in which the control stores the result
HANDLE	Q parameter with identification of the transaction

Note

When writing to tables, the control checks the lengths of the string parameters. If the entries exceed the length of the columns to be described, then the control outputs an error message.

Example

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM
   Tab_Example"
* - ...
31SQL INSERT Q1 HANDLE Q5
```

23.6.10 Example

In the following example, the defined material is read from the table (**WMAT.TAB**) and is stored as a text in a QS parameter. The following example shows a possible application and the necessary program steps.

> You can use the **FN 16** function, for example, in order to reuse QS parameters in your own log files.

Use synonym

0 BEGIN PGM SQL_READ_WMAT MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table-\WMAT.TAB'"	; Create synonym
2 SQL BIND QS1800 "my_table.WMAT"	; Bind QS parameters
3 SQL QL1 "SELECT WMAT FROM my_table WHERE NR==3"	; Define search
4 SQL FETCH Q1900 HANDLE QL1	; Execute search
5 SQL ROLLBACK Q1900 HANDLE QL1	; Complete transaction
6 SQL BIND QS1800	; Remove parameter binding
7 SQL Q1 "DROP SYNONYM my_table"	; Delete synonym
8 END PGM SQL_READ_WMAT MM	

Step	Explanation
1 Create synonym	Assign a synonym to a path (replace long paths with short names) ■ The path **TNC:\table\WMAT.TAB** is always placed in single quotes ■ The selected synonym is **my_table**
2 Bind QS parameters	Bind a QS parameter to a table column ■ **QS1800** is freely available in NC programs ■ The synonym replaces the entry of the complete path ■ The defined column from the table is called **WMAT**
3 Define search	A search definition contains the entry of the transfer value ■ The **QL1** local parameter (freely selectable) serves to identify the transaction (multiple transactions are possible simultaneously) ■ The synonym defines the table ■ The **WMAT** entry defines the table column of the read operation ■ The entries **NR** and **==3** define the table rows of the read operation ■ Selected table columns and rows define the cells of the read operation

4 Execute search	The control performs the read operation
	■ **SQL FETCH** copies the values from the **result set** into the bound Q or QS parameter
	  ■ **0** successful read operation
	  ■ **1** faulty read operation
	■ The syntax **HANDLE QL1** is the transaction designated by the parameter **QL1**
	■ The parameter **Q1900** is a return value for checking whether the data have been read

5 Complete transaction	The transaction is concluded and the used resources are released

Step	Explanation
6 Remove binding	The binding between table columns and QS parameters is removed (release of necessary resources)
7 Delete synonym	The synonym is deleted (release of necessary resources)

> Synonyms are an alternative only to the required absolute paths. Relative path entries are not possible.

The following NC program shows the entry of an absolute path.

0 BEGIN PGM SQL_READ_WMAT_2 MM	
1 SQL BIND QS 1800 "'TNC:\table-\WMAT.TAB'.WMAT"	; Bind QS parameters
2 SQL QL1 "SELECT WMAT FROM 'TNC:-\table\WMAT.TAB' WHERE NR ==3"	; Define search
3 SQL FETCH Q1900 HANDLE QL1	; Execute search
4 SQL ROLLBACK Q1900 HANDLE QL1	; Complete transaction
5 SQL BIND QS 1800	; Remove parameter binding
6 END PGM SQL_READ_WMAT_2 MM	

24

Graphical Programming

24.1 Fundamentals

Application

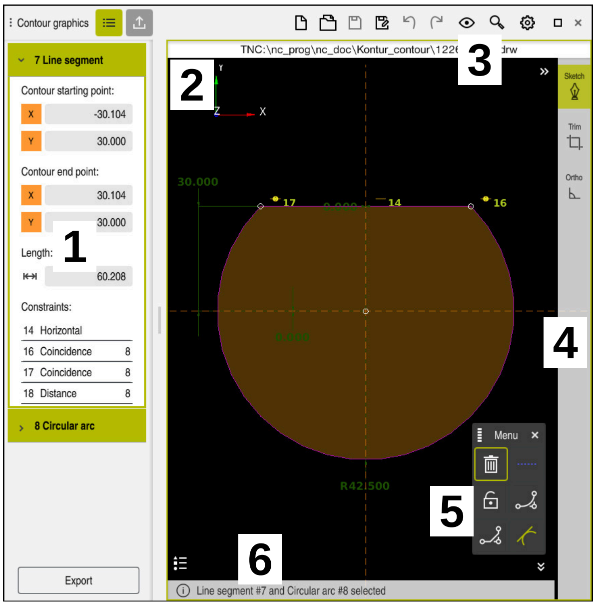
Graphical programming offers an alternative to conventional Klartext programming. You can create 2D sketches by drawing lines and arcs and generate a contour from this in Klartext. In addition, you can import existing contours from an NC program into the **Contour graphics** workspace and edit them graphically.

You can use graphical programming in isolation via a separate tab or in the form of the separate **Contour graphics** workspace. If you use graphical programming on its own tab, you cannot open any other workspaces of the **Editor** operating mode on this tab.

Description of function

The **Contour graphics** workspace is available in the **Editor** operating mode.

Screen layout



Screen layout of the **Contour graphics** workspace

The **Contour graphics** workspace contains the following areas:

1 Element information area
2 Drawing area
3 Title bar
4 Drawing functions
5 Toolbar
6 Information bar

Controls and gestures in graphical programming

In graphical programming, you can create a 2D sketch using various elements.

Further information: "First steps in graphical programming", Page 1352

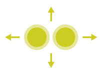
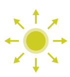
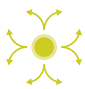
The following elements are available in graphical programming:

- Line segment
- Arc
- Construction point
- Construction line
- Construction circle
- Chamfer
- Rounding arc

Gestures

In addition to the gestures specifically available for graphical programming, you can also use various general gestures in graphical programming.

Further information: "Common gestures for the touchscreen", Page 96

Symbol	Gesture	Meaning
	Tap	Select a point or element
	Long press	Insert construction point
	Two-finger drag	Move the drawing view
	Draw straight elements	Insert **Line segment** element
	Draw circular elements	Insert **Circular arc** element

Icons of the title bar

The title bar of the **Contour graphics** workspace shows not only icons solely available for graphical programming but also general icons of the control interface.

Further information: "Icons on the control's user interface", Page 102

The control shows the following icons in the title bar:

Icon or shortcut	Meaning
	File options
CTRL+N	**Discard contour**
CTRL+O	**Open file**
	View settings
	Show dimensions
	Show restrictions
	Show reference axes
	Preset views menu
	Include defined drawing area With this function, the control shows the defined size of the drawing area. You can define the size of the drawing area in the contour settings. **Further information:** "Contour settings window", Page 1344
	Include selected element
	Include drawn elements in drawing area
	Open the **Contour settings** window **Further information:** "Contour settings window", Page 1344

Possible colors

The control shows the elements in the following colors:

Icon	Meaning
	Element A drawn element that is not fully dimensioned is shown by the control in orange as a solid line.
	Construction element Drawn elements can be converted to construction elements. You can use construction elements to obtain additional points for creating your sketch. Construction elements are shown by the control in blue as a dashed line.
	Reference axis The reference axes shown form a Cartesian coordinate system. Dimensioning in the contour editor starts from the intersection of the reference axes. The intersection of the reference axes corresponds to the workpiece preset when exporting the contour data. The control shows reference axes in brown as a dashed line.
	Locked element You cannot adjust locked elements. If you want to edit a locked element, you must unlock it first. Locked elements are shown by the control in red as a solid line.
	Fully dimensioned element The control shows fully dimensioned elements in dark green. You cannot attach any additional constraints or dimensions to a fully dimensioned element, otherwise the element will be over-determined.
	Contour element The control shows the contour elements between the **Start Point** and **End Point** in the **Export** menu as green solid elements.

Icons in the drawing area

The control shows the following icons in the drawing area:

Icon or shortcut	Designation	Meaning
	Milling direction	The selected **Milling direction** determines whether the defined contour elements are output clockwise or counterclockwise.
	Delete	Deletes all selected elements
	Change the annotation	Switches the display between length and angle dimensions.
	Toggle construction element	This function converts an element into a construction element. Construction elements cannot also be output when exporting a contour.
	Lock element	If this icon is displayed, the selected element is locked against editing. Select the icon to unlock the element.
	Unlock element	If this icon is displayed, the selected element is not locked against editing. Select the icon to lock the element.
	Set the datum	This function moves the selected point to the origin of the coordinate system. All other drawn elements are also moved according to the given distances and dimensions. If necessary, the **Set the datum** function recalculates the existing restrictions.
	Corner rounding	Inserts a rounding arc
	Chamfer	Inserts a chamfer
	Coincidence	This function sets the **Coincidence** constraint for two marked points. When you use this function, the selected points of two elements are connected together. "Coincidence" is used here to refer to these points coinciding.
	Vertical	This function sets the **Vertical** restriction for the selected **Line segment** element. Vertical elements are automatically vertical.
	Horizontal	This function sets the **Horizontal** restriction for the selected **Line segment** element. Horizontal elements are automatically horizontal.
	Perpendicular	This function sets the **Perpendicular** restriction for two selected elements of the type **Line segment**. There is an angle of 90° between perpendicular elements.

Icon or shortcut	Designation	Meaning
	Parallel	This function sets the **Parallel** restriction for two selected elements of the type **Line segment**. When you apply this function, the angle of two lines is aligned. First, the control checks whether there are restrictions, e.g. **Horizontal**. Behavior in the case of restrictions: ■ If there is a restriction, the **Line segment** without restriction is aligned with the **Line segment** with restriction. ■ If both lines have restrictions, the function cannot be applied. The dimension is over-determined. ■ If there are no restrictions, the order of selection is decisive. The **Line segment** selected in the second instance is aligned with the **Line segment** first selected.
=	**Equal**	This function sets the **Equal** constraint for two marked elements. When you apply this function, the sizes of two elements are matched, e.g. in length or diameter of two elements. First, the control checks whether there are restrictions, e.g. a defined length. Behavior in the case of restrictions: ■ If there is a restriction, the element without restriction is aligned with the element with restriction. ■ If both elements have corresponding restrictions, the function cannot be applied. The dimension is over-determined. ■ If there are no restrictions, the control generates the average value from the given **Circular arc** size values.
	Tangential	This function sets the **Tangential** restriction for two marked elements of the type **Line segment** and **Circular arc** or **Circular arc** and **Circular arc**. When you use this function, both arcs and lines are moved. The affected elements come into contact at exactly one point after they are moved and form a tangential transition.
	Symmetry	This function sets the **Symmetry** restriction for a marked element of the type **Line segment** and two marked points of other construction elements. When you apply this function, the control positions the distance of the two points symmetrically to the selected line. If you subsequently change the distance of one of the points, the other point automatically adjusts to the change.
	Point on element	This function sets the **Point on element** restriction for a selected element and a point of another selected element. When you apply this function, the selected point is moved to the selected element.
	Legend	Use this function to show or hide the legend explaining all the controls.
CTRL+D	**Sketch**	To prevent you from unintentionally drawing elements while moving the drawing, you can deactivate drawing mode. Drawing mode remains disabled until you activate it again. If you deactivate drawing mode, the control changes the button to green.

Icon or shortcut	Designation	Meaning
CTRL+T	**Trim**	If multiple elements overlap, you can use **Trim** mode to shorten elements to the next adjacent element. **Trim** mode remains active until you deactivate it again. If the function is active, the control changes the button to green.
	Ortho	With this function, you can only draw rectangular lines. The control does not allow oblique lines or arcs. If the function is active, the control changes the button to green.
CTRL+A	**Select all**	The **Select All** function allows you to mark all drawn elements at once.

Contour settings window

The **Contour settings** window contains the following areas:

- **General information**
- **Sketching**
- **Export**

General information area

The **General information** area contains the following settings:

Setting	Meaning
Plane	You select the plane in which you want to draw by selecting an axis combination. Available planes: ■ **XY** ■ **ZX** ■ **YZ**
Diameter programming	You use a switch to select whether drawn rotation contours in the XZ and YZ planes are interpreted as radius or diameter dimensions during export.
Sketching area width	Default width of the drawing area
Sketching area height	Default height of the drawing area
Decimal places	Number of decimal places for dimensioning

Sketching area

The **Sketching** area contains the following settings:

Setting	Meaning
Rounding radius	Default size for an inserted rounding radius
Chamfer length	Default size for an inserted chamfer
Snap circle size	Size of the snap circle when selecting the elements

Export area

The **Export** area contains the following settings:

Setting	Meaning
Type of circle	You select whether arcs are output as **CC** and **C** or **CR**.
Export as RND	You use a switch to select whether roundings drawn with the **RND** function are also exported as **RND** to the NC program.
CHF output	You use a switch to select whether chamfers drawn with the **CHF** function are also exported as **CHF** to the NC program.

24.1.1 Creating a new contour

To create a new contour:

- Select the **Editor** operating mode
- Select **Add**
- The control opens the **Quick selection** and the **Open File** workspaces.
- Select **New contour**
- The control opens the contour in a new tab.

24.1.2 Locking and unlocking elements

If you want to protect an element from editing, you can lock the element. A locked element cannot be edited. If you want to edit the locked element, you must first unlock the element.

To lock and unlock elements in graphical programming:

- Select the drawn element
- Select the **Lock element** function
- The control locks the element.
- The control displays the locked element in red.
- Select the **Unlock element** function
- The control unlocks the element.
- The control displays the unlocked element in yellow.

Notes

- Set the **Contour settings** before drawing.
  Further information: "Contour settings window", Page 1344
- Dimension each element immediately after drawing. If you do not dimension until the entire contour has been drawn, the contour may move unintentionally.
- You can assign restrictions to the drawn elements. To avoid unnecessarily complicating the design, work only with necessary restrictions.
  Further information: "Icons in the drawing area", Page 1342
- If you select elements of the contour, the control turns the elements in the menu bar green.

Definitions

File type	Definition
H	NC program in Klartext format
TNCDRW	HEIDENHAIN contour file

24.2 Importing contours into graphical programming

Application

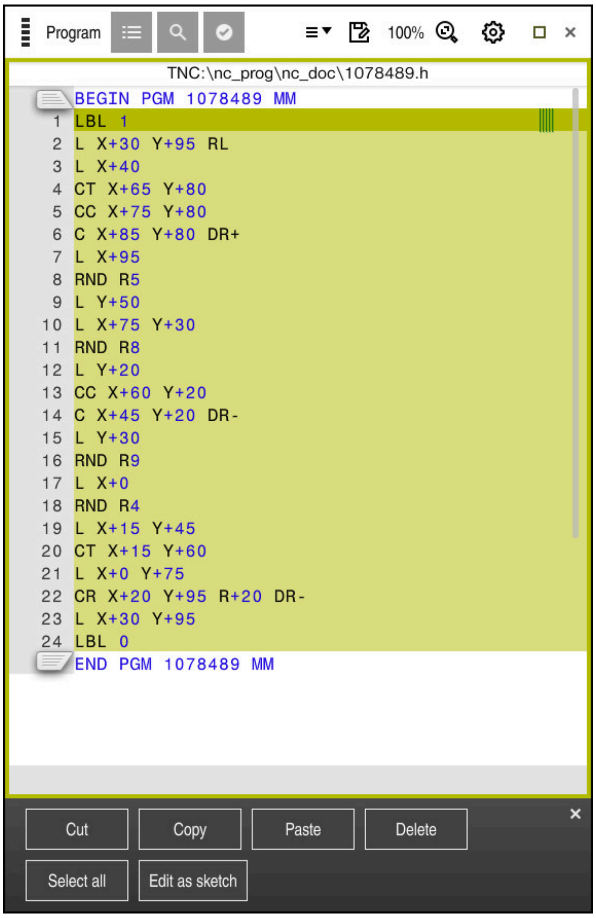
With the **Contour graphics** workspace, you can not only create new contours, but also import contours from existing NC programs and, if necessary, edit them graphically.

Requirements

- Max. 200 NC blocks
- No cycles
- No approach and retraction movements
- No straight lines **LN** (option 9)
- No technology data, e.g. feed rates or additional functions
- No axis motions that are outside the specified plane, e.g. XY plane

If you try to import a prohibited NC block into graphical programming, the control will issue an error message.

Description of function



Contour to be imported from the NC program

In graphical programming, all contours consist exclusively of linear or circular elements with absolute Cartesian coordinates.

The control converts the following path functions when importing to the **Contour graphics** workspace:

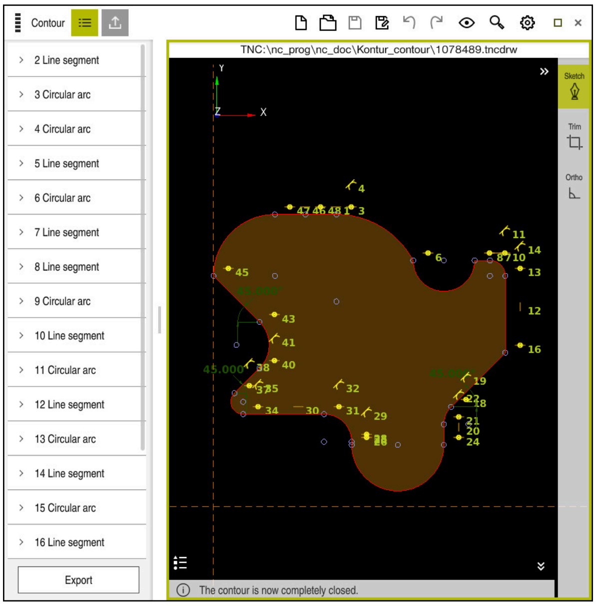
- Circular contour **CT**

  Further information: "Circular path CT", Page 303
- NC blocks with polar coordinates

  Further information: "Polar coordinates", Page 290
- NC blocks with incremental inputs

  Further information: "Incremental entries", Page 292
- Free contour programming **FK**

24.2.1 Importing contours



Imported contour

To import contours from NC programs:

- Select the **Editor** operating mode
- Open an existing NC program with a contour included
- Search for the contour in the NC program
- Hold the first NC block of the contour
- > The control opens the context menu.
- Select **Mark**
- > The control shows two marker arrows.
- Select the desired area with the marker arrows
- Select **Edit as sketch**
- > The control opens the selected contour area in the **Contour graphics** workspace.

Notes

- In the **Contour settings** window, you can specify whether the dimensions of rotational contours in the XZ plane or YZ plane are interpreted as radius or diameter dimensions.

  Further information: "Contour settings window", Page 1344
- When you use the **Edit as sketch** function to import a contour into graphical programming, all elements are initially locked. Before you begin editing the elements, you must unlock the elements.

  Further information: "Locking and unlocking elements", Page 1345
- You can edit contours graphically and export them after importing.

  Further information: "First steps in graphical programming", Page 1352

  Further information: "Exporting contours from graphical programming", Page 1349

24.3 Exporting contours from graphical programming

Application

The **Export** column in the **Contour graphics** workspace allows you to export newly created or graphically edited contours.

Related topics

- Importing contours

  Further information: "Importing contours into graphical programming", Page 1346
- First steps in graphical programming

  Further information: "First steps in graphical programming", Page 1352

Description of function

The **Export** column provides the following functions:

- **Contour starting point**

  Use this function to define the **Contour starting point** of the contour. You can either set the **Contour starting point** graphically or enter an axis value. If you enter an axis value, the control automatically determines the second axis value.
- **Contour end point**

  Use this function to define the **Contour end point** of the contour. You can set the **Contour end point** in the same way as the **Contour starting point**.
- **Invert direction**

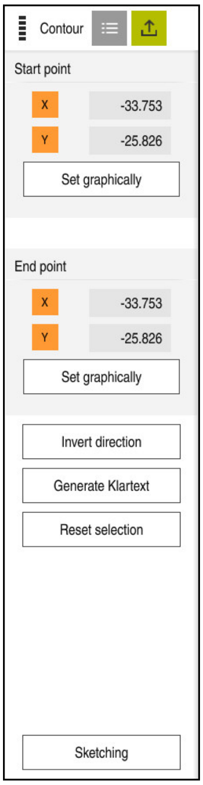
  Use this function to change the programming direction of the contour.
- **Generate Klartext**

  Use this function to export the contour as an NC program or subprogram. The control can only export certain path functions. All generated contours contain absolute Cartesian coordinates.

  Further information: "Contour settings window", Page 1344

  The contour editor can generate the following path functions:
  - Line **L**
  - Circle center **CC**
  - Circular contour **C**
  - Circular contour **CR**
  - Radius **RND**
  - Chamfer **CHF**
- **Reset selection**

  Use this function to deselect a contour.

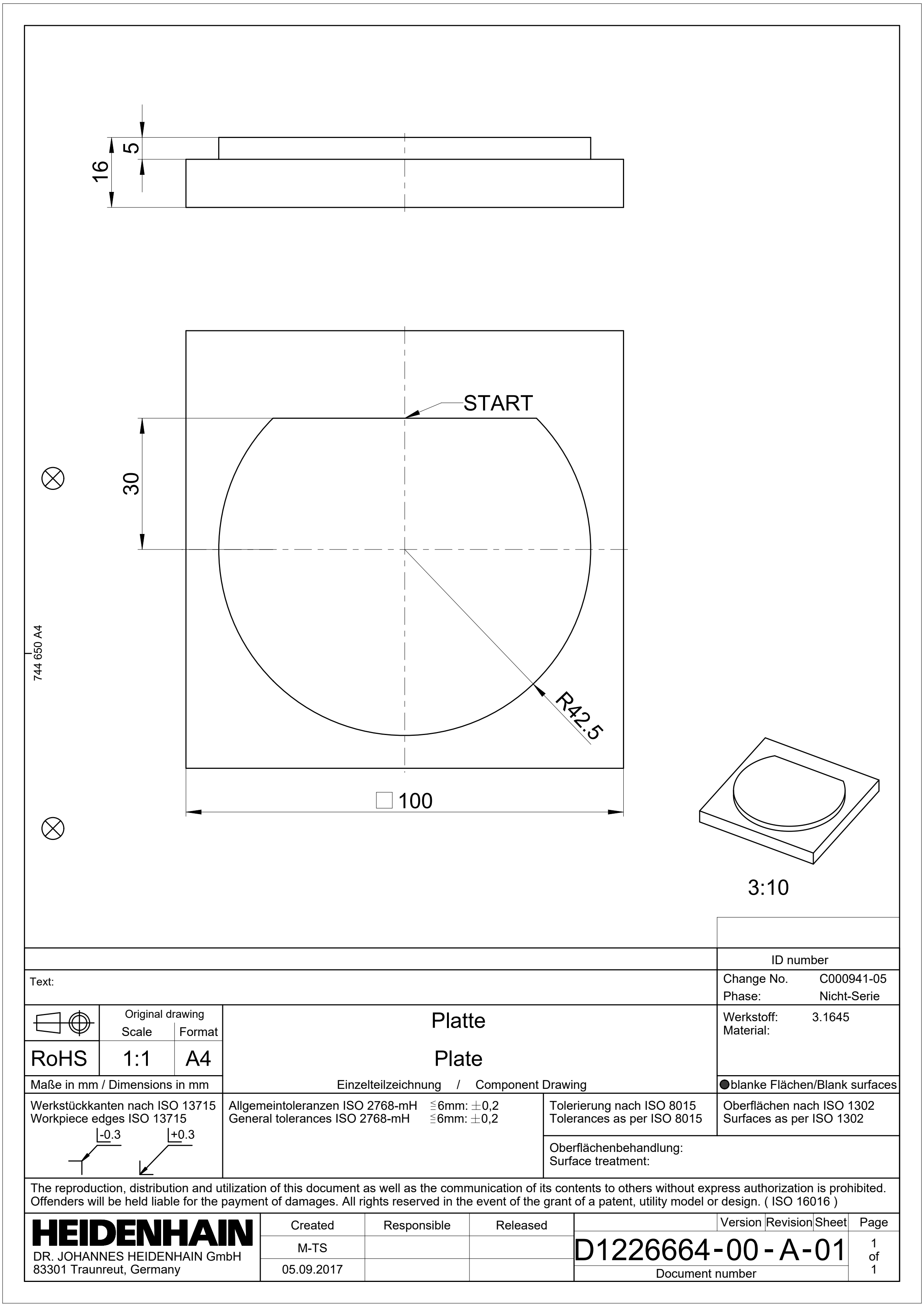


Notes

- You can also use the **Contour starting point** and **Contour end point** functions to pick up parts of the drawn elements and generate a contour from them.
- You can save drawn contours with the file type ***.tncdrw** to the control.

24.4 First steps in graphical programming

24.4.1 Example task D1226664



24.4.2 Drawing a sample contour

To draw the displayed contour:

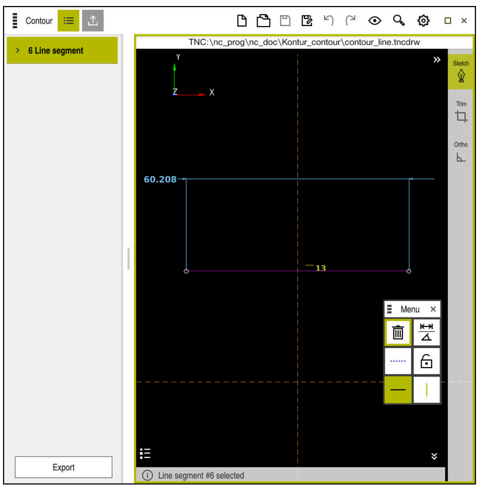
- Create a new contour

  Further information: "Creating a new contour", Page 1345
- Configure **Contour settings**

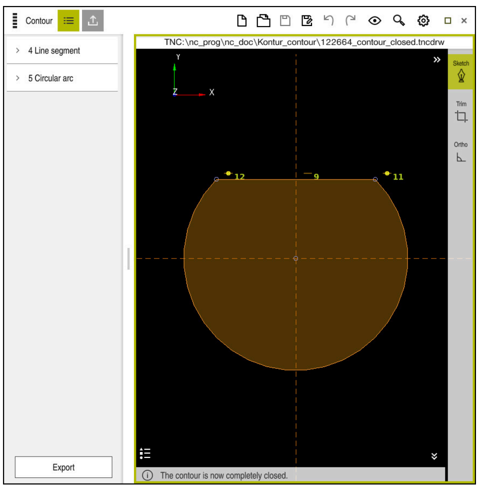
> In the **Contour settings** window, you can define basic settings for drawing. For this example, you can use the standard settings.

Further information: "Contour settings window", Page 1344

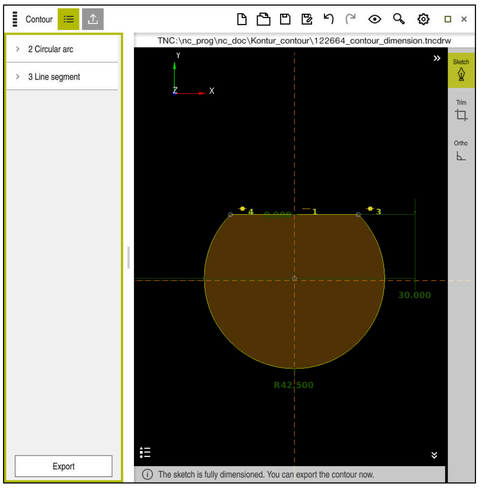
- Draw horizontal **Line segment**
- Select the end point of the drawn line
- > The control shows the X and Y distance of the line to the center.
- Enter Y distance to center, e.g. **30**
- > The control positions the line according to the condition set.
- Draw **Circular arc** from one end point of the line to the other end point
- > The control displays the closed contour in yellow.
- Select the center point of the arc
- > The control shows the center point coordinates of the arc in **X** and **Y**.
- Enter **0** for the X and Y center point coordinates of the arc
- > The control moves the contour.
- Select drawn arc
- > The control shows the current radius value of the arc.
- Enter radius **42.5**
- > The control adjusts the radius of the arc.
- > The contour is fully defined.



Line drawn



Closed contour



Dimensioned contour

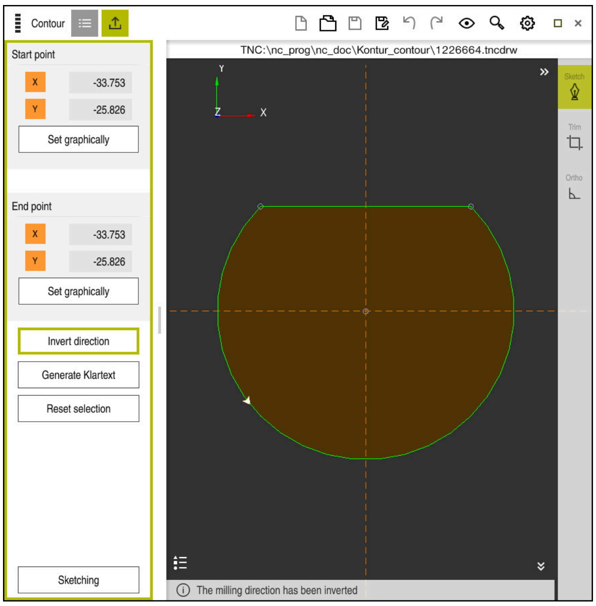
24.4.3 Exporting a drawn contour

To export the drawn contour:

- Draw contour
- Select the **Export** column
- > The control displays the **Export** column.
- Select **Set graphically** in the **Contour starting point** area
- Select the starting point on the drawn contour
- > The control shows the coordinates of the selected start point, the selected contour and the programming direction.

> You can adjust the programming direction of the contour with the **Invert direction** function.

- Select the **Generate Klartext** function
- > The control generates the contour based on the defined data.



Selected contour elements in the **Export** column with defined **Milling direction**

25

Opening CAD Files with the CAD-Viewer

25.1 Fundamentals

Application

CAD-Viewer enables you to open following standardized CAD data formats directly on the control:

File	Type	Format
Step	.STP and .STEP	■ AP 203 ■ AP 214
IGES	.IGS and .IGES	■ Version 5.3
DXF	.DXF	■ R10 to 2015
STL	STL	■ Binary ■ ASCII

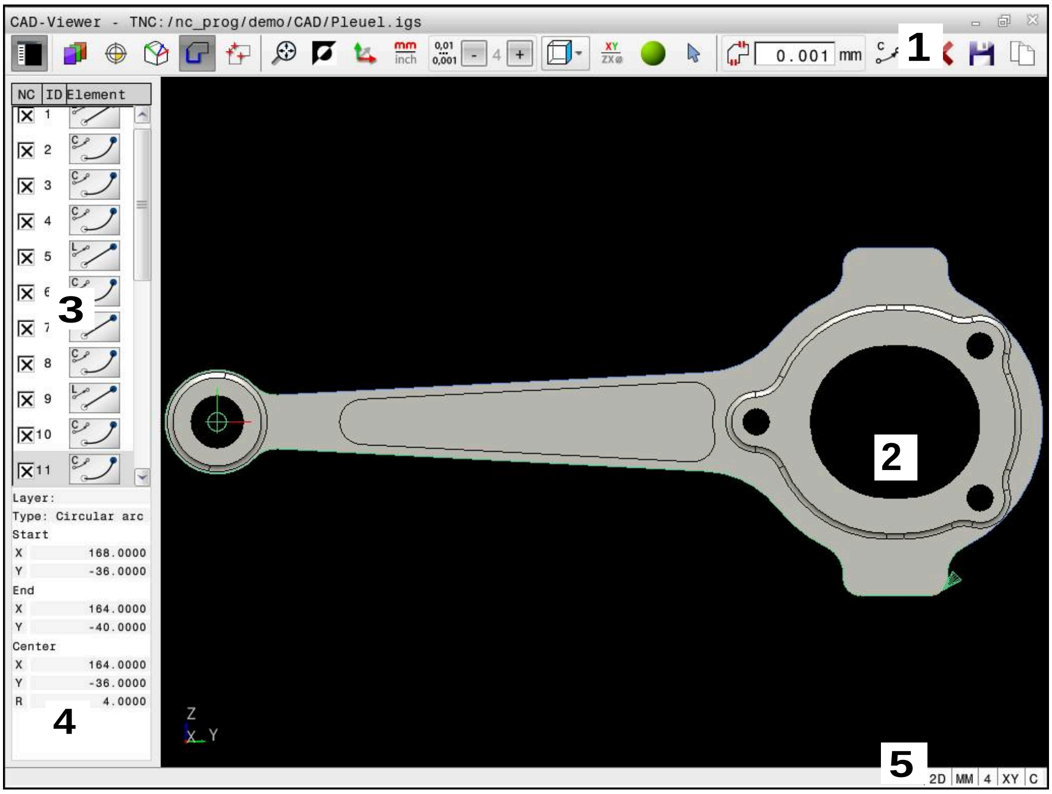
The **CAD-Viewer** runs as a separate application on the third desktop of the control.

Related topics

- Creating 2D sketches on the control
  Further information: "Graphical Programming", Page 1337

Description of function

Screen layout



CAD file open in **CAD-Viewer**

CAD Viewer consists of the following areas:

1 Menu bar
Further information: "Menu bar icons", Page 1357
2 Graphics window
The CAD model is displayed in the graphics window.
3 Sidebar window
The Sidebar window displays information on the active function, e.g. available layers or the position of the workpiece preset.
4 Element Information window
Further information: "Element Information window", Page 1359
5 Status bar
The status bar contains the active settings.

Menu bar icons

The menu bar contains the following icons:

Icon	Function
	**Show sidebar** Show or hide the Sidebar window
	**Display the layer** Display the layer(s) in the Sidebar window **Further information:** "Layer", Page 1360

Icon	Function
	Preset Define the workpiece preset Workpiece preset has been defined Delete the defined workpiece preset **Further information:** "Workpiece preset in the CAD model", Page 1361
	Datum Set the datum The datum has been set **Further information:** "Workpiece datum in the CAD model", Page 1364
	Contour Select contour (option 42) **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1366
	Positions Select drilling positions (option 42) **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1366
	3D mesh Create a 3D mesh (option 152) **Further information:** "Generating STL files with 3D mesh (option 152)", Page 1372
	Show all Set the zoom to the largest possible view of the complete graphics
	Inverted colors Change the background color (black or white)
	Toggle between 2D and 3D modes
mm inch	Set the unit of measure (mm or inches) of the output. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1366
0,01 ... 0,001	**Number of decimal places** Select the resolution. The resolution defines the number of decimal places and the number of positions for linearization. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1366 Default setting: 4 decimal places with **mm**, and 5 decimal places with **inch** as the unit of measure
	Set perspective Switch between various views of the model e.g. **Top**

Icon	Function
XY	**Axes** Select the working plane: ■ **XY** ■ **YZ** ■ **ZX** ■ **ZXØ** In the **ZXØ** working plane, you can select turning contours (option 50). If you take over a contour or position, the control will output the NC program in the selected working plane. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1366
	Toggle a 3D model between a solid model and a wire-frame model.
	"Select, add, or remove contour elements" mode ℹ The icon shows the current mode. Clicking the icon activates the next mode. **Further information:** "Applying contours and positions to NC programs with CAD Import (option 42)", Page 1366
	Undo

Element Information window

In the Element Information window, the following information is displayed for the selected element of the CAD file:

- Associated layer
- Element type
- Point type:
  - Point coordinates
- Line type:
  - Coordinates of the starting point
  - Coordinates of the end point
- Circular arc or circle type:
  - Coordinates of the starting point
  - Coordinates of the end point
  - Coordinates of the center point
  - Radius

Layer

CAD files usually contain several layers. The designer uses these layers to create groups of various types of elements, e.g. the actual workpiece contour, dimensions, auxiliary and design lines, shadings, and texts.

The CAD file to be processed must contain at least one layer. Elements not assigned to a layer are automatically moved by the control to the "anonymous" layer.

With the **Display the layer** icon, the control shows all the layers of the file in the List view window. Use the checkbox in front of the name to show and hide the individual layers.

When you open a CAD file in the **CAD-Viewer**, all layers available are shown.

If you hide unnecessary layers, the graphic becomes clearer.

Notes

- The control does not support binary DXF format. Save the DXF file in ASCII format in the CAD or drawing program.
- Before loading the file into the control, ensure that the name of the file contains only permitted characters.

  Further information: "Permitted characters", Page 1070
- When you select a layer in the list view window, you can use the space bar to show and hide the layer.

25.2 Workpiece preset in the CAD model

Application

The datum of the drawing in the CAD file is not always located in a manner that lets you use it as a workpiece preset. Therefore, the control provides a function with which you can shift the workpiece preset to a suitable location by clicking an element. You can also define the orientation of the coordinate system.

Related topics

- Presets in the machine

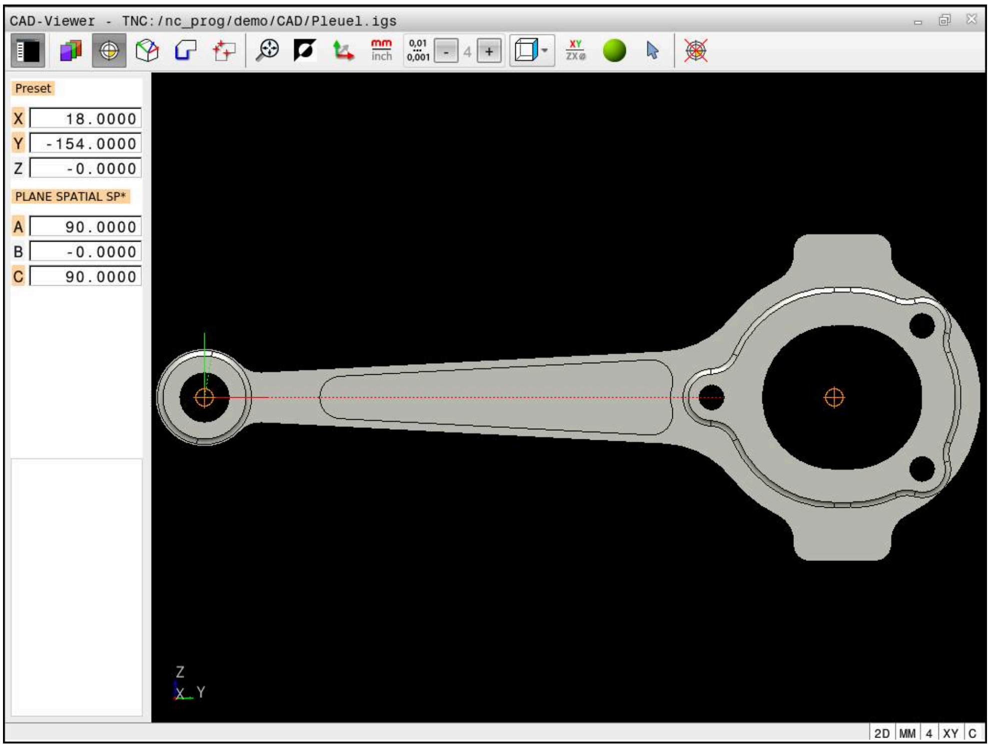
  Further information: "Presets in the machine", Page 187

Description of function

When you select the **Preset** icon, the control displays the following information in the list view window:

- Distance between the defined preset and the drawing datum
- Orientation of the coordinate system with respect to the drawing

The control displays values not equal to 0 in orange.



Workpiece preset in the CAD model

You can position the preset at the following locations:

- By directly inputting numerical values into the List View window
- For straight lines:
  - Starting point
  - Center
  - End point
- For circular arcs:
  - Starting point
  - Center
  - End point
- For full circles:
  - At the quadrant transitions
  - At the center
- At the intersection between:
  - Two straight lines, even if the point of intersection is actually on the extension of one of the lines
  - Straight line and circular arc
  - Straight line and full circle
  - Two circles (regardless of whether a circular arc or a full circle)

If you have set a workpiece preset, the control displays the **Preset** icon in the menu bar with a yellow quadrant.

The preset and optional orientation are inserted in the NC program as a comment starting with **origin**.

```
4 ;orgin = X... Y... Z...
5 ;orgin_plane_spatial = SPA... SPB... SPC...
```

You can change the preset even after you have selected the contour. The control does not calculate the actual contour data until you save the selected contour in a contour program.

25.2.1 Set the workpiece preset or workpiece datum and align the coordinate system

- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.

  Further information: "Common gestures for the touchscreen", Page 96
- The following contents also apply to the workpiece datum. In this case, select the **Datum** icon to begin with.

Set workpiece preset or workpiece datum on individual element

To set the workpiece preset on an individual element:

- Select **Preset**
- Position the cursor on the desired element
- > If you are using a mouse, the control for the element displays selectable presets using gray icons.
- Click the icon at the desired position
- > The control sets the workpiece preset to the selected position. The control turns the icon green.
- Align the coordinate system, if required

Set the workpiece preset or workpiece datum at the intersection of two elements

You can set the workpiece preset at intersection points of straight lines, full circles and arcs.

To set the workpiece preset at the intersection of two elements:

- ▶ Select **Preset**
- ▶ Click on the first element
- \> The control highlights the element in color.
- ▶ Click on the second element
- \> The control sets the workpiece preset at the intersection of the two elements. The control marks the workpiece preset with a green icon.
- ▶ Align the coordinate system, if required

> - If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
> - If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
> - If the control cannot calculate an intersection, it deselects the previously selected element.

Aligning the coordinate system

The following conditions must be met in order to align the coordinate system:

- Preset has been defined
- There are elements next to the preset that can be used for the desired alignment

To align the coordinate system:

- ▶ Select an element in the positive direction of the X axis
- \> The control aligns the X axis.
- \> The control changes the angle **C** in the list view window.
- ▶ Select an element in the positive direction of the Y axis
- \> The control aligns the Y and Z axes.
- \> The control changes the angles **A** and **C** in the list view window.

25.3 Workpiece datum in the CAD model

Application

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control has a function with which you can define a new datum and a tilting operation.

Related topics

- Presets in the machine

  Further information: "Presets in the machine", Page 187

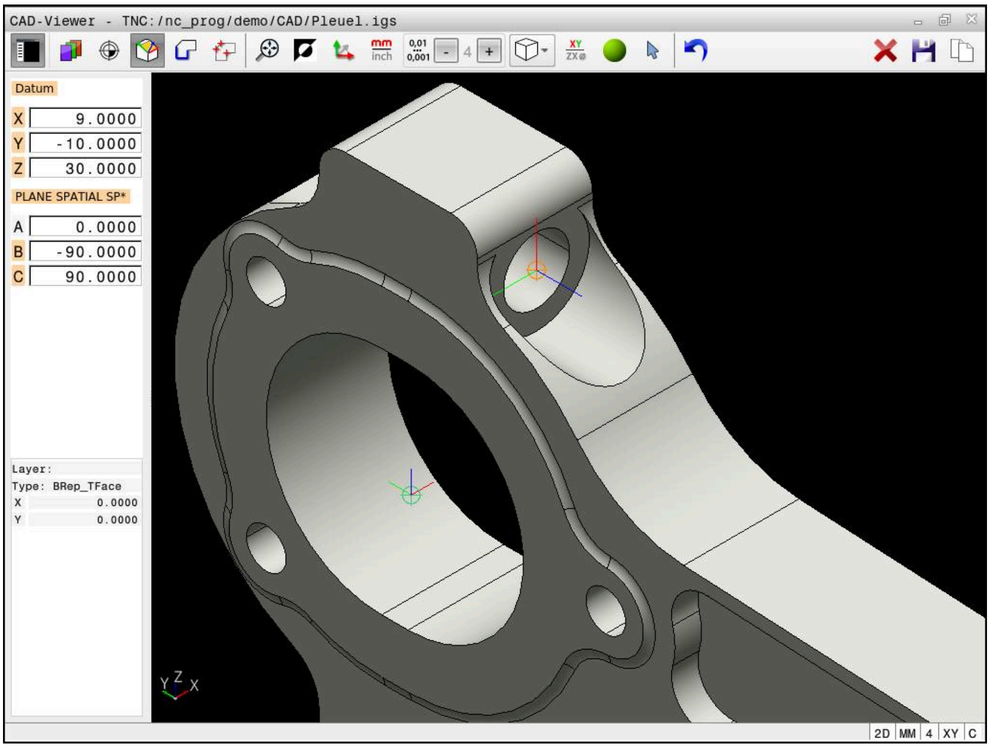
Description of function

When you select the **Datum** icon, the control displays the following information in the list view window:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the coordinate system

You can set a workpiece datum and also move it further by entering values directly in the list view window.

The control displays values not equal to 0 in orange.



Workpiece datum for tilted machining

The datum with the orientation of the coordinate system can be set at the same positions as a preset.

Further information: "Workpiece preset in the CAD model", Page 1361

If you have set a workpiece datum, the control displays the **Datum** icon in the menu bar with a yellow area.

Further information: "Set the workpiece preset or workpiece datum and align the coordinate system", Page 1363

The datum and its optional orientation can be inserted as NC block or comments in the NC program by using the **TRANS DATUM AXIS** function for the datum and the **PLANE SPATIAL** function for the orientation.

If you define only one datum and its orientation, then the control inserts the functions in the NC program as an NC block.

```
4 TRANS DATUM AXIS X... Y... Z...
5 PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

If you additionally select contours or points, then the control inserts the functions in the NC program as comments.

```
4 ;TRANS DATUM AXIS X... Y... Z...
5 ;PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

25.4 Applying contours and positions to NC programs with CAD Import (option 42)

Application

You can open CAD files directly on the control in order to extract contours and machining positions from it. You can then store them as Klartext programs or as point files. Klartext programs acquired in this manner can also be run on older HEIDENHAIN controls, since these contour programs contain only **L** and **CC**/**C** blocks.

Related topics

- Using point tables

  Further information: "Point tables", Page 346

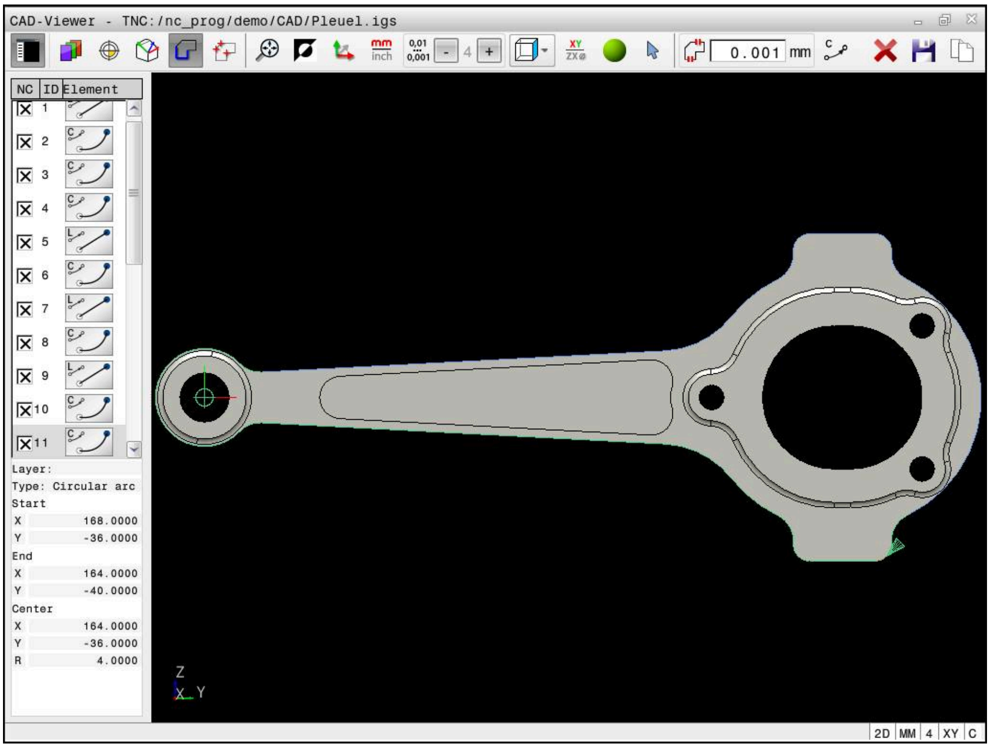
Requirement

- CAD Import (software option 42)

Description of function

To insert a selected contour or a selected machining position directly into an NC program, use the control's clipboard. Using the clipboard, you can even transfer the contents to the software tools (e.g., **Leafpad** or **Gnumeric**).

Further information: "Opening files with additional software", Page 1972



CAD model with marked contour

Icons in the CAD Import

With the CAD Import, the control shows the following additional functions in the menu bar:

Icon	Function
	Delete entire list
	Save entire list content to a file
	Copy entire list contents to clipboard
	Set the transition tolerance The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred when the drawing was made. The default setting is 0.001 mm
	C or **CR** Arc mode defines whether circular arcs are output in C format or CR format (e.g., for cylinder surface interpolation) in the NC program.
	Show connections between two positions Specifies whether the control should display the tool path as a dashed line during the selection of machining positions
	Apply path optimization The control optimizes the tool traverse movement to produce shorter traverse movements between the machining positions. When the icon is pressed again, the optimization is reset
	Find circles according to diameter range. Load center coordinates to the position list The control opens a pop-up window in which you can filter holes (full circles) based on their size

Applying contours

The following elements can be selected as a contour:

- Line segment
- Circle
- Circular arc
- Polyline
- Any curves (e.g., splines, ellipses)

You can also use the CAD viewer (option 50) to select contours for turning. The icon is grayed out if option 50 is not enabled. Before selecting a turning contour, you must set the preset on the rotary axis. If you select a turning contour, it is saved with Z and X coordinates. In addition, all X coordinate values in turning contours are transferred as diameter values, i.e. the drawing dimensions for the X axis are doubled. All contour elements below the rotary axis cannot be selected and are highlighted gray.

Linearization

During linearization, a contour is divided into individual positions. The CAD Import creates a straight line **L** for each position. With the CAD Import, you can therefore also apply contours that cannot be programmed with the path functions of the control, e.g. splines.

The **CAD-Viewer** linearizes all of the contours that are not in the XY plane. The finer the resolution, the more accurately the control will display the contours.

Applying positions

You can also use the CAD Import to save positions, e.g. for holes.

Three possibilities are available in the pattern generator for defining machining positions:

- Single selection
- Multiple selection within a range
- Multiple selection using search filters

Further information: "Select positions", Page 1370

The following file types are available:

- Point table (**.PNT**)
- Klartext conversational language program (**.H**)

If you save the machining positions to a Klartext program, the control creates a separate linear block with a cycle call for every machining position (**L X... Y... Z... F MAX M99**).

Multi-selection filter settings

After you have used the quick selection function to mark hole positions, a pop-up window appears in which the smallest diameter found is to the left and the largest diameter to the right. With the buttons just below the diameter display you can adjust the diameter so that you can load the hole diameters that you want.

The following buttons are available:

Icon	Filter setting of smallest diameter
|<<	Display the smallest diameter found (default setting)
<	Display the next smaller diameter found
>	Display the next larger diameter found
>>	Display the largest diameter found. The control sets the filter for the smallest diameter to the value set for the largest diameter

Icon	Filter setting of largest diameter
<<	Display the smallest diameter found. The control sets the filter for the largest diameter to the value set for the smallest diameter
<	Display the next smaller diameter found
>	Display the next larger diameter found
>>|	Display the largest diameter found (default setting)

25.4.1 Selecting and saving a contour

- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.

  Further information: "Common gestures for the touchscreen", Page 96
- Deselecting, deleting, and saving of elements works in the same way for taking over contours and positions.

Selecting a contour with existing contour elements

To select and save a contour with existing contour elements:

- ▶ Select **Contour**
- ▶ Place the cursor on the first contour element
- > The control shows the suggested direction of rotation as a dashed line.
- ▶ If necessary, move the cursor towards the more distant end point.
- > The control changes the suggested direction of rotation.
- ▶ Select the contour element
- > The selected contour element is displayed in blue and is marked in the Sidebar window.
- > Other contour elements are shown in green.

The control suggests the contour that deviates least from the suggested direction. To change the suggested contour path, you can select paths independently of the existing contour elements

- ▶ Select the last desired contour element
- > All contour elements up to the selected element are shown in blue and are marked in the Sidebar window.

- ▶ Select **Save entire list content to a file**
- > The **Define file name for contour program** window opens.
- ▶ Enter the desired name
- ▶ Select the path to the storage location
- ▶ Select **Save**
- > The selected contour is saved as an NC program.

- Alternatively, you can use the **Copy entire list contents to clipboard** icon to copy the selected contour to the clipboard and then paste it into an existing NC program.
- If you select an element with the CTRL key pressed, it is deselected for export.

Selecting paths independent of existing contour elements

To select a path independent of existing contour elements:

- Select **Contour**

- Select **Select**
- > The icon changes, and the control activates the **Add** mode.
- Place the cursor relative to the desired contour element
- > The control displays selectable points:
  - End point or center point of a line or curve
  - Quadrant transitions or center of a circle
  - Points of intersection between existing elements
- Select the desired point
- Select more contour elements

> If the contour element to be extended or shortened is a straight line, the control will extend or shorten the contour element along the same line. If the contour element to be extended or shortened is a circular arc, the control will extend or shorten the contour element along the same arc.

Saving the contour as a workpiece blank definition (option 50)

For a workpiece blank definition in turning mode, a closed contour is required.

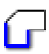
NOTICE

Danger of collision!

Closed contours must completely lie inside the workpiece blank definition. Otherwise, the system will follow closed contours also along the rotary axis when machining, causing collisions.

- Select or program only those contour elements that are actually required, e.g. within the definition of a finished part.

To select a closed contour:



- Select **Contour**
- Select all required contour elements
- Select the starting point of the first element
- > The control closes the contour.

25.4.2 Select positions

> - The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.
>
>   **Further information:** "Common gestures for the touchscreen", Page 96
> - Deselecting, deleting, and saving of elements works in the same way for taking over contours and positions.
>
>   "Selecting and saving a contour"

Individual selection

To select individual positions, e.g. holes:

- ▶ Select **Positions**
- ▶ Position the cursor on the desired element
- > The control shows the circumference and center point of the element in orange.
- ▶ Select the desired element
- > The control highlights the selected element in blue and displays it in the list view window.

Multiple selection by area

To select multiple positions within an area:

- ▶ Select **Positions**
- ▶ Select **Select**
- > The icon changes, and the control activates the **Add** mode.
- ▶ Drag a box around the area while holding down the left mouse button
- > The control opens the **Find circle centers after diameter range** window and shows the smallest and largest diameter found.
- ▶ Change the filter settings as needed
- ▶ Press **OK**
- > The control highlights all positions of the selected diameter range in blue and shows them in the list view window.
- > The control shows the traversing distance between the positions.

Multiple selection by search filter

To select multiple positions using a search filter:

- ▶ Select **Positions**
- ▶ **Find circles according to diameter range.** Select **Find circles according to diameter range. Load center coordinates to the position list**
- > The control opens the **Find circle centers after diameter range** window and shows the smallest and largest diameter found.
- ▶ Change the filter settings as needed
- ▶ Press **OK**
- > The control highlights all positions of the selected diameter range in blue and shows them in the list view window.
- > The control shows the traversing distance between the positions.

Notes

- Set the correct unit of measure, because the CAD file does not contain this information.
- Ensure that the unit of measure used in the NC program matches with that used in the **CAD-Viewer**. Elements that have been copied from the **CAD-Viewer** to the clipboard do not contain any information about the unit of measure.
- The control also transfers two workpiece-blank definitions (**BLK FORM**) to the contour program. The first definition contains the dimensions of the entire CAD file. The second one, which is the active one, contains only the selected contour elements, so that an optimized size of the workpiece blank results.

Notes on applying contours

- If you double-click a layer in the list view window, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of a contour.
- Select the first contour element such that approach without collision is possible.
- You can even select a contour if the designer has saved the lines on different layers.
- Specify the direction of rotation during contour selection so that it matches the desired machining direction.
- The contour paths available depend on the selectable contour elements that are shown in green. Without the green elements, the control will display all solutions available. To remove the proposed contour path, select the first green element by pressing the left mouse button while holding the **CTRL** key down.
  As an alternative, you can switch to the Remove mode:

25.5 Generating STL files with 3D mesh (option 152)

Application

With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective files of fixtures and tool holders, for example, or to position STL files generated from the simulation for another machining operation.

Related topics

- Fixture Monitoring (option 40)
  Further information: "Fixture monitoring (option 40)", Page 1091
- Export simulated workpiece as STL file
  Further information: "Exporting a simulated workpiece as STL file", Page 1415
- Using an STL file as workpiece blank
  Further information: "Defining a workpiece blank with BLK FORM", Page 234

Requirement

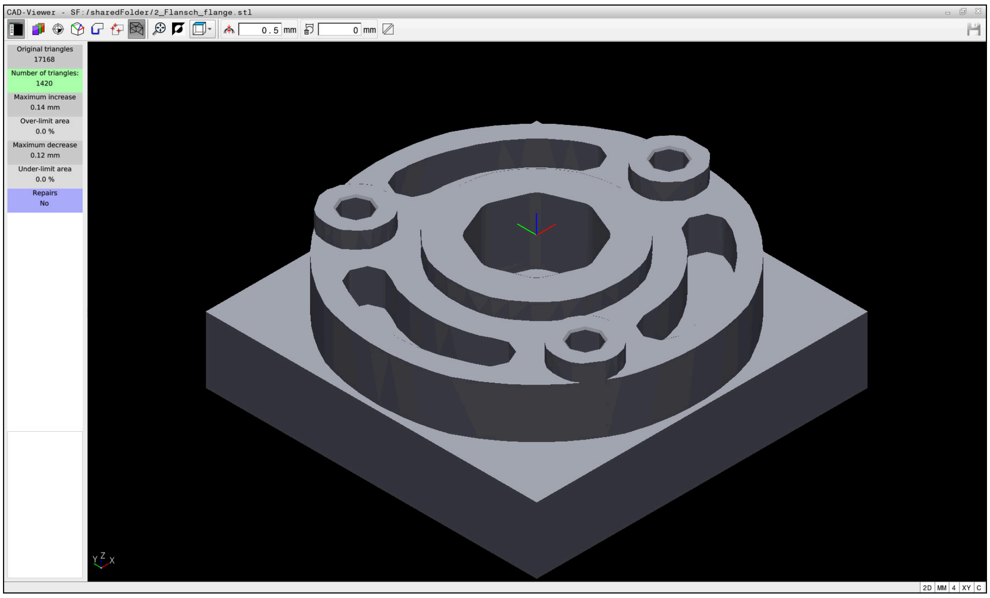
- Software option 152, CAD Model Optimizer

Description of function

When you select the **3D mesh** icon, the control changes to **3D mesh** mode. The control covers the 3D model displayed in **CAD-Viewer** with a mesh of triangles.

The control simplifies the original model and removes errors, e.g. small holes in a solid or self-intersections of a surface.

You can save the result and use it for various control functions, e.g. as a workpiece blank with the **BLK FORM FILE** function.



3D model in **3D mesh** mode

The simplified model or parts of it may be smaller or larger than the original model. The result depends on the quality of the original model and the selected settings in **3D mesh** mode.

The Sidebar window shows the following information:

Option	Meaning
Original triangles	Number of triangles in the original model
Number of triangles:	Number of triangles with active settings in the simplified model If this option is highlighted in green, the number of triangles is in the optimum range. You can further reduce the number of triangles using the available functions. **Further information:** "Functions for the simplified model", Page 1374
Maximum increase	Maximum increase of the triangle mesh
Over-limit area	Surface increase in percent compared to the original model
Maximum decrease	Maximum decrease of the triangle mesh compared to the original model
Under-limit area	Surface decrease in percent compared to the original model

Option	Meaning
Repairs	Indicates whether the original model has been repaired or not If it has been repaired, the control will indicate the type of repair, e.g. **Yes: Hole Int Shells**. This indication consists of the following items: ■ **Hole** **CAD-Viewer** closed holes in the 3D model. ■ **Int** **CAD-Viewer** removed self-intersections. ■ **Shells** **CAD-Viewer** joined multiple separate solids.

In order to use STL files for control functions, the saved files must meet the following requirements:

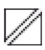
- Max. 20 000 triangles
- Triangular mesh forms a closed shell

The greater the number of triangles in an STL file, the greater the processing power required by the control for simulation.

Functions for the simplified model

In order to reduce the number of triangles, you can define further settings for the simplified model.

CAD-Viewer provides the following functions:

Symbol	Function
	**Allowed simplification** Use this function to simplify the output model by the specified tolerance. The higher the value, the more the surfaces may deviate from the original.
	**Remove holes <= diameter** Use this function to remove holes and pockets up to the specified diameter from the original model.
	**Only optimized mesh shown** To be able to evaluate the deviations, use this function to toggle superimposition of the view containing the optimized triangle mesh with the original mesh from the original file.
	**Save** Use this function to save the simplified 3D model with the selected settings as an STL file.

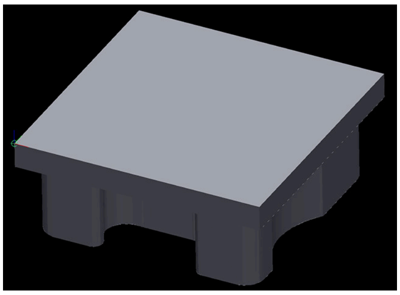
25.5.1 Positioning the 3D model for rear-face machining

To position an STL file for rear-face machining:

- Export the simulated workpiece as an STL file

  Further information: "Saving a simulated workpiece as STL file", Page 1417

- Select the **Files** operating mode
- Select the exported STL file
- > The control opens the CAD files in **CAD-Viewer**.
- Select **Preset**
- > In the Sidebar window, the control displays information on the position of the preset.
- Enter the value of the new preset under **Preset**, e.g. **Z-40**
- Confirm your input
- Orient the coordinate system by specifying values under **PLANE SPATIAL SP***, e.g. **A+180** and **C+90**
- Confirm your input



- Select **3D mesh**
- > The control opens the **3D mesh** mode and simplifies the 3D model using the default settings.
- Further simplify the 3D model using the **3D mesh** mode functions, if required.

  Further information: "Functions for the simplified model", Page 1374

- Select **Save**
- > The control opens the **Define file name for 3D mesh** menu.
- Enter the desired name
- Select **Save**
- > The control saves the STL file positioned for rear-face machining.

> The resulting file can then be used for rear-face machining with the **BLK FORM FILE** function.
>
> **Further information:** "Defining a workpiece blank with BLK FORM", Page 234

26

User Aids

26.1 Help workspace

Application

In the **Help** workspace the control displays a help graphic for the current syntax element of an NC function or the **TNCguide** integrated product help.

Related topics

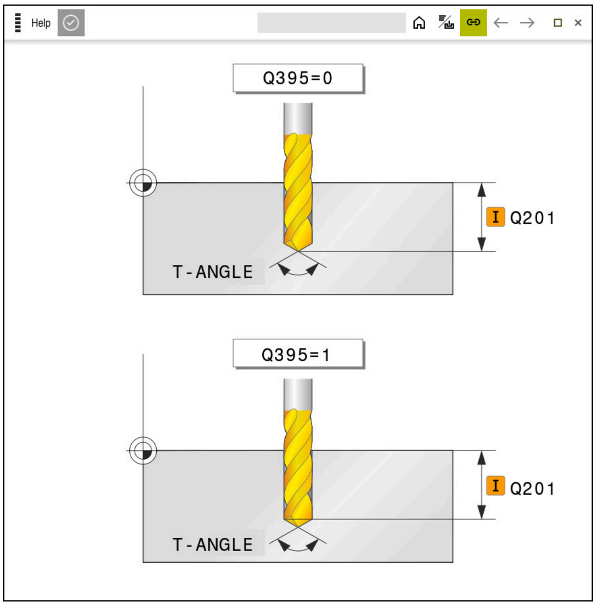
- **Help** application
  Further information: "Help application", Page 63
- User's Manual as the **TNCguide** integrated product help
  Further information: "User's Manual as integrated product help: TNCguide", Page 62

Description of function

The **Help** workspace can be selected in the **Editor** operating mode and in the **MDI** application.

Further information: "Editor operating mode", Page 192

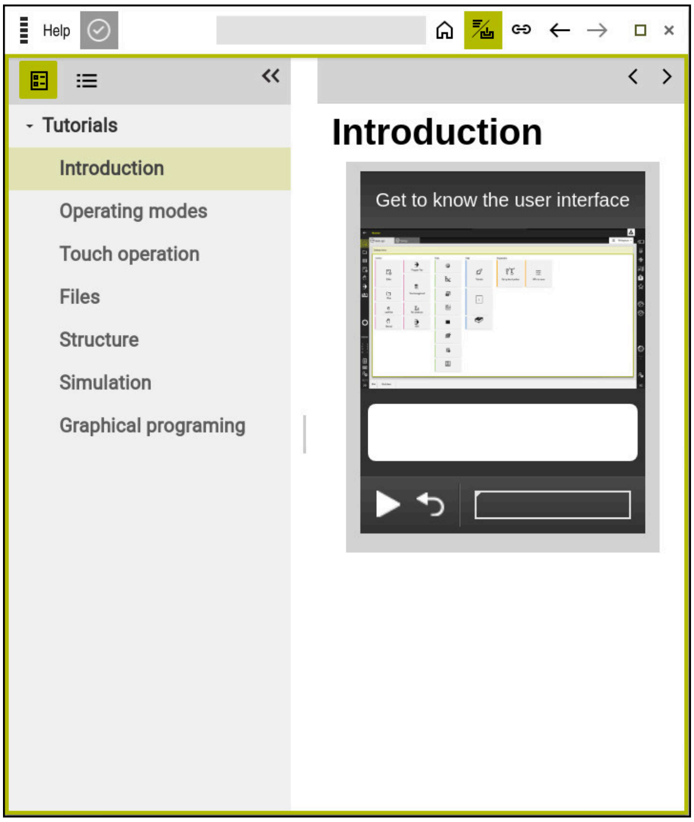
Further information: "Application MDI", Page 1759



Help workspace with a help graphic for a cycle parameter

If the **Help** workspace is active, the control can display the help screen in it during programming instead of in the **Program** workspace.

Further information: "Program workspace", Page 194



Help workspace with opened **TNCguide**

If the **Help** workspace is active, the control can display the integrated **TNCguide** product help.

Further information: "User's Manual as integrated product help: TNCguide", Page 62

Symbols in the Help workspace

Symbol	Function
	Show start page The start page displays all available documentation. Select the desired documentation, using a navigation tiles, e.g. **TNCguide**. If only one piece of documentation is available, the control opens the content directly. When a documentation is open, you can use the search function. **Further information:** "Symbols", Page 64
	Displaying the **TNCguide** **Further information:** "User's Manual as integrated product help: TNCguide", Page 62
	Displaying help images during programming

26.2 Virtual keyboard of the control bar

Application

You can use the virtual keyboard for entering NC functions, letters, and numbers, and for navigation.

The virtual keyboard offers the following modes:

- NC input
- Text input
- Formula entry

Description of function

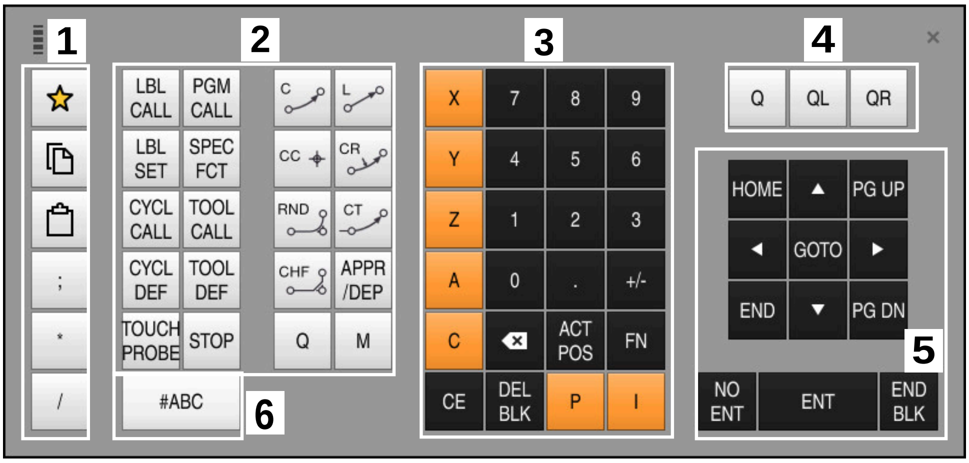
The control opens NC input mode by default after the start procedure.

You can move the keyboard on the screen. The keyboard remains active, even when the operating mode is switched, until the keyboard is closed.

The control remembers the position and mode of the virtual keyboard until it is shut down.

The **Keyboard** workspace provides the same functions as the virtual keyboard.

NC input areas



Virtual keyboard in NC input mode

NC input mode contains the following areas:

1 File functions
- Define favorites
- Copy
- Paste
- Add comment
- Add structure item
- Hide NC block

2 NC functions

3 Axis keys and numerical input

4 Q parameters

5 Navigation and dialog keys

6 Switch to text input

> If you press the **Q** button in the NC functions area repeatedly, the control cycles through the syntax in the following sequence:
> - **Q**
> - **QL**
> - **QR**

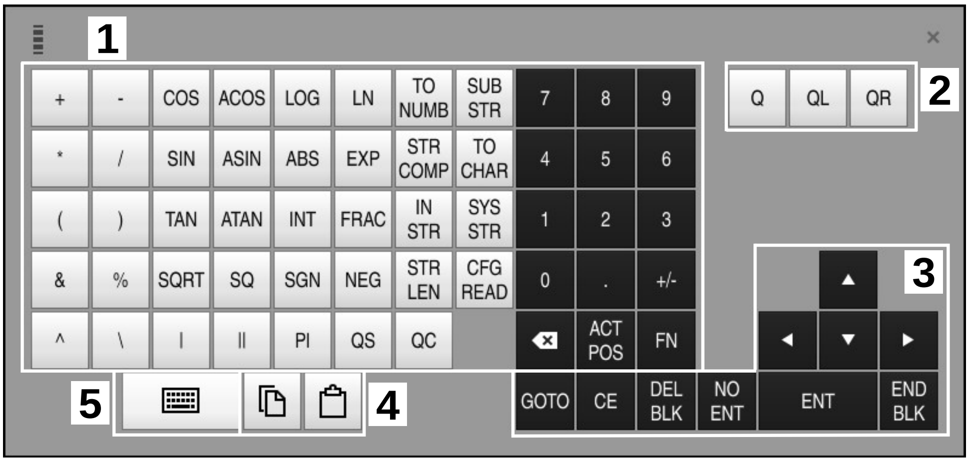
Areas of text input

Virtual keyboard in text input mode

The text input contains the following areas:

1 Input
2 Navigation and dialog keys
3 Copying and pasting
4 Switch to formula input

Areas of formula input



Virtual keyboard in formula input mode

The formula input contains the following areas:

1 Input
2 Q parameters
3 Navigation and dialog keys
4 Copying and pasting
5 Switch to NC input

26.2.1 Opening and closing the virtual keyboard

To open the virtual keyboard:

- ▶ Select the **virtual keyboard** on the control bar
- > The control opens the virtual keyboard.

To close the virtual keyboard:

- ▶ Select the **virtual keyboard** when the virtual keyboard is open
- ▶ Or press **Close** in the virtual keyboard
- > The control closes the virtual keyboard.

26.3 GOTO function

Application

With the **GOTO** key or the **GOTO block number** button you define an NC block at which the control positions the cursor. In the **Tables** mode you use the **GOTO record** button to define a table row.

Description of function

If an NC program is open for simulation or execution, the control additionally positions the execution cursor in front of the NC block. The control then starts program run or the simulation beginning from the defined NC block without considering the preceding lines of the NC program.

You can enter the block number directly or find it in the NC program with the **Search** function.

26.3.1 Selecting an NC block with GOTO

To select an NC block:

- ▶ Select **GOTO**
- > The control opens the **GOTO jump instruction** window.
- ▶ Enter the block number
- ▶ Press **OK**
- > The control positions the cursor to the defined NC block.

NOTICE

Danger of collision!

If you select an NC block in program run using the **GOTO** function and then execute the NC program, the control ignores all previously programmed NC functions, e.g. transformations. This means that there is a risk of collision during subsequent traversing movements!

- ▶ Use **GOTO** only when programming and testing NC programs
- ▶ Only use **Block scan** when executing NC programs

Further information: "Block scan for mid-program startup", Page 1786

Notes

- You can also use the keyboard shortcut **CTRL+G** instead of the **GOTO** button.
- If the control in the action bar shows an icon for selection, you can open the selection window with **GOTO**.

26.4 Adding comments

Application

You can add comments to an NC program in order to explain program steps or make general notes.

Description of function

You have the following possibilities for adding comments:

- Comment within an NC block
- Comment as a separate NC block
- Define existing NC block as comment

The control marks comments with a preceding **;** character. The control does not execute comments during simulation or program run.

A comment may contain up to 255 characters.

> The last character in a comment block must not be a tilde (**~**).

26.4.1 Adding a comment as an NC block

To add a comment as a separate NC block:

- Select the NC block after which the comment is to be added

;

- Select **;**
- > After the selected NC block, the control adds a comment as a new NC block.
- Define the comment

26.4.2 Adding a comment in an NC block

To add a comment within an NC block:

- Edit the desired NC block

;

- Select **;**
- > The control inserts a **;** character at the end of the block.
- Define the comment

26.4.3 Commenting an NC block out or in

Use the **Comment out/in** button to define an existing NC block as a comment or to change a comment back to an NC block.

To comment an existing NC block in or out:

- ▶ Select the desired NC block

- ▶ Select **Comment Off/On**
- > The control inserts a **;** character at the beginning of the block.
- > If the NC block is already defined as a comment, the control removes the **;** character.

26.5 Hiding NC blocks

Application

Use **/** or the **Hide/Show** button to hide NC blocks.

If you hide NC blocks, you can skip the hidden NC blocks in program run.

Related topics

- **Program Run** operating mode

  Further information: "Program Run operating mode", Page 1778

Description of function

If you mark an NC block with a **/** character, then the NC block is hidden. If you activate the **Skip /** switch in the **Program Run** operating mode or in the **MDI** application, the control skips these NC blocks during program run.

Further information: "Icons and buttons", Page 1780

26.5.1 Hiding or showing NC blocks

To hide or show an NC block:

- ▶ Select the desired NC block

- ▶ Select **Skip Off/On**
- > The control adds a **/** character before the NC block.
- > If the NC block is already hidden, the control removes the **/** character.

26.6 Structuring of NC programs

Application

You can use structure items to make long and complex NC programs more clear and legible, and also to navigate more quickly through an NC program.

Related topics

- **Structure** column of the **Program** workspace

  Further information: "Structure column in the Program workspace", Page 1386

Description of function

You can use structure items to arrange your NC programs. Structure items are texts that you can use as comments or headlines for the subsequent program lines.

A structure item may contain up to 255 characters.

The control displays the structure items in the **Structure** column.

Further information: "Structure column in the Program workspace", Page 1386

26.6.1 Adding a structure item

To insert a structure item:

- Select the NC block after which you want to add the structure item

*

- Select *
- After the selected NC block, the control adds a structure item as a new NC block.
- Define the structure text

26.7 Structure column in the Program workspace

Application

When you open an NC program, the control searches the NC program for structure elements and displays these structure elements in the **Structure** column. The structure elements act like links and thus enable fast navigation in the NC program.

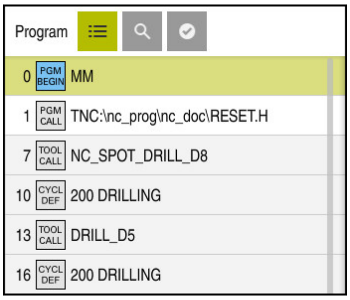
Related topics

- **Program** workspace, defining contents of the **Structure** column

  Further information: "Settings in the Program workspace", Page 197
- Inserting structure items manually

  Further information: "Structuring of NC programs", Page 1385

Description of function



Structure column with automatically created structure elements

When you open an NC program, the control automatically creates the structure.

In the **Program settings** window, you define which structure elements the control displays in the structure.

Further information: "Settings in the Program workspace", Page 197

The **Structure** column shows the following information:

- NC block number
- Icon of the NC function
- Function-dependent information

The control displays the following icons within the structure:

Icon	Syntax	Information
PGM BEGIN	**BEGIN PGM**	Unit of measurement of the NC program **MM** or **INCH**
TOOL CALL	**TOOL CALL**	Depending on the name selected in the **TOOL CALL**: ■ Tool name ■ Number of the tool If you do not specify a name or number in the **TOOL CALL**, the control does not display any additional information.
*	*** Structure block**	String entered in the NC block
LBL SET	**LBL SET**	Depending on the name selected in the dialog: ■ Name of the label ■ Number of the label
LBL SET	**LBL 0**	Label number 0
CYCL DEF	**CYCL DEF**	Number and name of the defined cycle
TCH PROBE	**TCH PROBE**	Number and name of the defined cycle
MON START	**MONITORING SECTION START**	String entered in the **AS** syntax element
MON STOP	**MONITORING SECTION STOP**	No additional information
PGM CALL	**PGM CALL**	Path of the called NC program, e.g. **TNC:\Safe.h**

Icon	Syntax	Information
SPEC FCT	**FUNCTION MODE**	Selected machining mode **MILL** or **TURN**
STOP M0	**STOP** or **M0**	No additional information
M1	**M1**	No additional information
M2 M30	**M2** or **M30**	No additional information

26.7.1 Editing an NC block using the structure

To edit an NC block using the structure:

- Open an NC program
- Open the **Structure** column
- Select structure element
- > The control positions the cursor on the corresponding NC block in the NC program. The focus of the cursor remains in the **Structure** column.
- Select the right arrow
- > The focus of the cursor changes to the NC block.
- Select the right arrow
- > The control edits the NC block.

Notes

- In the case of long NC programs, establishing the structure may take longer than loading the NC program. Even if the structure has not yet been created, you can still work independently of it in the loaded NC program.
- You can navigate within the **Structure** column using the up and down arrow keys.
- The control shows called NC programs in the structure with a white background. If you double-tap or click on such a structure element, the control opens the NC program if necessary in a new tab. When the NC program is open, the control switches to the corresponding tab.

26.8 Search column in the Program workspace

Application

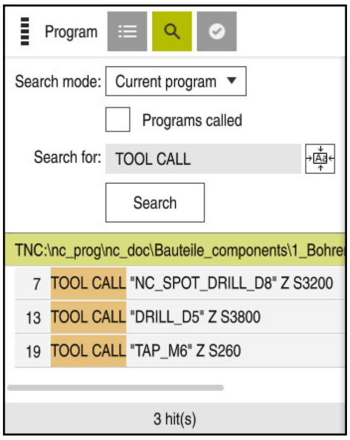
In the **Search** column, you can search the NC program for any character strings, e.g. individual syntax elements. The control lists all the results found.

Related topics

- Search for the same syntax element in the NC program with the arrow keys

  Further information: "Searching for the same syntax elements in different NC blocks", Page 201

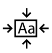
Description of function



Search column in the **Program** workspace

The control only provides the full range of functions in the **Editor** operating mode. In the **MDI** application and the **Program Run** operating mode, you can only search in the active NC program.

The control provides the following functions, icons and buttons in the **Search** column:

Area	Function
Search in:	■ **Current program** Search the current NC program and optionally all called NC programs ■ **Opened programs** Browse all open NC programs ■ **Search and replace** Search for strings and replace them with new strings, e.g. syntax elements **Further information:** "Search and replace mode", Page 1390
Search for:	In the input area, you define the search term. If you have not yet entered any characters, the control provides the last six search terms to choose from.
	Use the **Apply selection** icon to transfer the currently selected syntax element to the input area. If the selected NC block is not edited, the control accepts the syntax opener.
Search	Use this button to start the search in the **Current program** and **Opened programs** modes.

The control shows the following information about the results:

- Number of results
- File paths of the NC programs
- NC block numbers
- Entire NC blocks

The control groups the results according to NC programs. If you select a result, the control positions the cursor on the corresponding NC block.

Search and replace mode

In **Search and replace** mode, you can search for strings and replace the results found with other strings, e.g. syntax elements.

The control performs a syntax check before replacing a syntax element. With the syntax check, the control ensures that the new content results in correct syntax. If the result produces a syntax error, the control does not replace the content and displays a message.

In **Search and replace** mode, the control provides the following checkboxes and buttons:

Checkbox or button	Meaning
Search backward	The control searches the NC program from bottom to top.
Wrap around	The control searches the entire NC program, beyond the start and end of the NC program.
Find next	The control searches the NC program for the search term. The control marks the next result in the NC program.
Replace	The control performs a syntax check and replaces the selected content in the NC program with the content of the **Replace with:** field.
Replace and find next	If a search has not yet been performed, the control only marks the first result. When a result is highlighted, the control performs a syntax check and automatically replaces the found content with the contents of the **Replace with:** field. The control then marks the next result.
Replace all	The control performs a syntax check and automatically replaces all found results with the contents of the **Replace with:** field.

26.8.1 Search for and replace syntax elements

To search for and replace syntax elements in the NC program:

- ▶ Select an operating mode, e.g. **Editor**
- ▶ Select the desired NC program
- > The control opens the selected NC program in the **Program** workspace.
- ▶ Open the **Search** column
- ▶ In the **Search in:** field, select the **Search and replace** function
- > The control displays the **Search for:** and **Replace with:** fields.
- ▶ In the **Search for:** field, enter the search content, e.g. **M4**
- ▶ In the **Replace with:** field, enter the desired content, e.g. **M3**
- ▶ Select **Find next**
- > The control marks the first result in the NC program.
- ▶ Select **Replace**
- > The control performs a syntax check and replaces the content if the check is successful.

Notes

- The search results are retained until you shut down the control or search again.
- If you double-tap or click on a search result in a called NC program, the control opens the NC program (on a new tab if not already open). If the NC program is already open, the control switches to the corresponding tab.

26.9 Program comparison

Application

Use the **Program comparison** function to determine differences between two NC programs. You can transfer the deviations to the active NC program. If there are unsaved changes in the active NC program, you can compare the NC program with the last saved version.

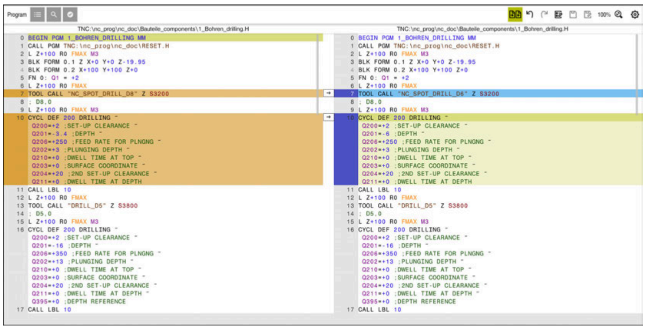
Requirements

- Max. 30,000 lines per NC program

  The control takes into account the actual lines, not the number of NC blocks. NC blocks can also contain several lines with one block number, e.g. cycles.

  Further information: "Contents of an NC program", Page 190

Description of function



Program comparison of two NC programs

You can use the program comparison only in the **Editor** operating mode in the **Program** workspace.

The control shows the active NC program on the right and the comparison program on the left.

The control marks differences with the following colors:

Color	Syntax element
Gray	Missing NC block or missing line for NC functions of different length
Orange	NC block with difference in comparison program
Blue	NC block with difference in the active NC program

During the program comparison, you can edit the active NC program, but not the comparison program.

If NC blocks differ, you can use an arrow symbol to transfer the NC blocks of the comparison program to the active NC program.

26.9.1 Applying differences to the active NC program

To transfer differences to the active NC program:

- Select the **Editor** operating mode
- Open an NC program
- Select **Program comparison**
- > The control opens a pop-up window for file selection.
- Select comparison program
- Select **Select**
- > The control shows both NC programs in the comparison view and marks all differing NC blocks.
- Select the arrow symbol for the desired NC block
- > The control transfers the NC block to the active NC program.
- Select **Program comparison**
- > The control closes the comparison view and transfers the differences to the active NC program.

Notes

- If the compared NC programs contain more than 1000 differences, the control cancels the comparison.
- If an NC program contains unsaved changes, the control displays an asterisk in front of the name of the NC program in the tab of the application bar.

26.10 Context menu

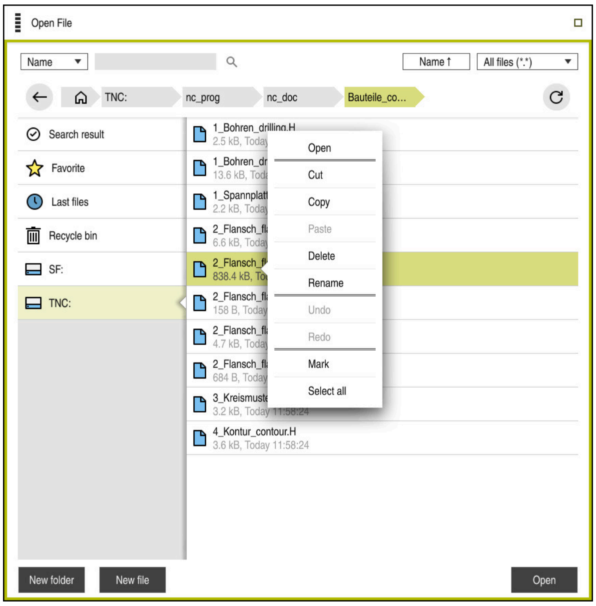
Application

With a long-press gesture or by right-clicking with the mouse, the control opens a context menu for the selected element, such as an NC block or file. Use the various functions of the context menu to run commands that affect the currently selected element(s).

Description of function

The functions available in the context menu depend on the selected element as well as the selected operating mode.

General



Context menu in the **Open File** workspace

The context menu offers the following functions:

- **Cut**
- **Copy**
- **Paste**
- **Delete**
- **Undo**
- **Redo**
- **Mark**
- **Select all**

> If you select the **Mark** or **Select all** functions, the control opens the action bar. The action bar displays all functions that are currently available for selection from the context menu.

As an alternative to the context menu, you can use keyboard shortcuts:

Further information: "Icons on the control's user interface", Page 102

Key or keyboard shortcut	Meaning
CTRL+BLANK	Mark the selected line
SHIFT+↑	Additionally mark a line above it
SHIFT+↓	Additionally mark a line below it
ESC	Cancel marking

> The keyboard shortcuts do not work in the **Job list** workspace.

Context menu in the Files operating mode

In the **Files** operating mode, the context menu also offers the following functions:

- **Open**
- **Select in Program Run**
- **Rename**

For the navigation functions, the context menu offers the respectively relevant functions, such as **Discard search results**.

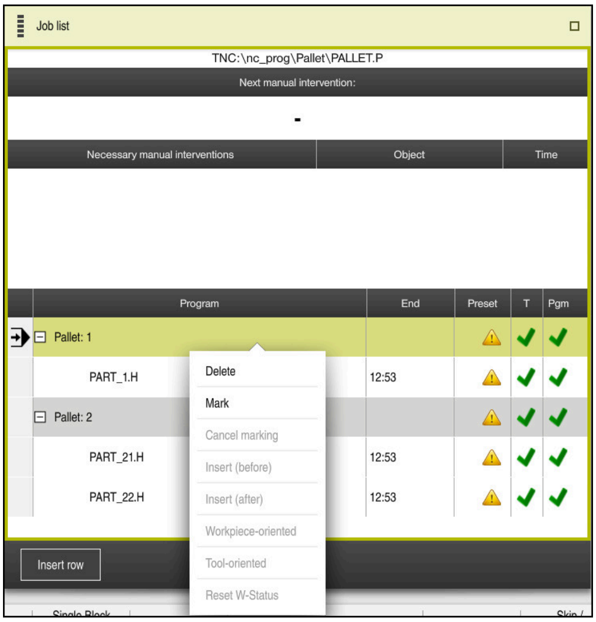
Further information: "Context menu", Page 1392

Context menu in the Tables operating mode

In the **Tables** operating mode the context menu additionally offers the **Cancel** function. Use the **Cancel** function to abort the marking action.

Further information: "Tables operating mode", Page 1802

Context menu in the Job list (option 22) workspace



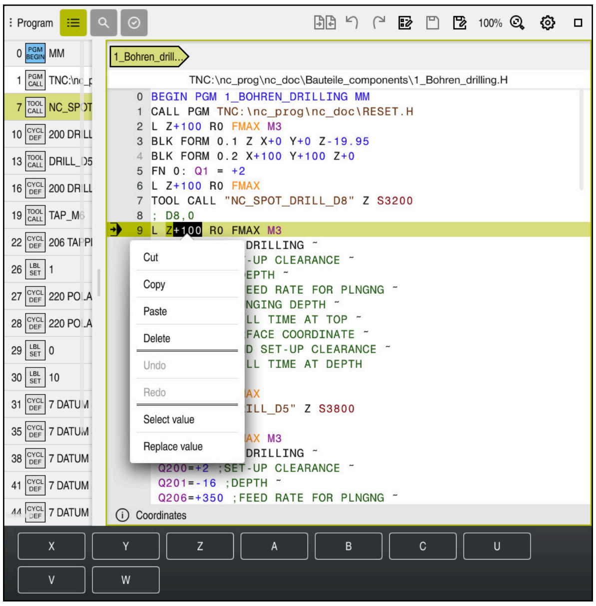
Context menu in the **Job list** workspace

In the **Job list** workspace, the context menu offers the following additional functions:

- **Cancel marking**
- **Insert (before)**
- **Insert (after)**
- **Workpiece-oriented**
- **Tool-oriented**
- **Reset W-Status**

Further information: "Job list workspace", Page 1764

Context menu in the Program workspace



Context menu for selected value in the **Program** workspace of the **Editor** operating mode

In the **Program** workspace, the context menu offers the following additional functions:

- **Edit as sketch**

  Only in the **Editor** operating mode

  Further information: "Importing contours into graphical programming", Page 1346
- **Select value**

  Active when you select a value of an NC block.
- **Replace value**

  Active when you select a value of an NC block.

Further information: "Program workspace", Page 194

> The **Select value** and **Replace value** functions are only available in the **Editor** operating mode and the **MDI** application.
>
> **Replace value** is also available during editing. In this case the otherwise necessary marking of the value to be replaced is omitted.
>
> For example, you can copy values from the calculator or position display to the clipboard and then paste them with the **Replace value** function.

Further information: "Calculator", Page 1396

Further information: "Status overview on the control bar", Page 147

If you select an NC block, the control displays marker arrows at the beginning and end of the selected area. Use these marker arrows to change the highlighted area.

Context menu in the configuration editor

In the configuration editor, the context menu also provides the following functions:

- **Direct entry of values**
- **Create copy**
- **Restore copy**
- **Change key name**
- **Open element**
- **Remove element**

Further information: "Machine parameters", Page 1958

26.11 Calculator

Application

The control offers a calculator on the control bar. You can copy the result to the clipboard and also paste values from the clipboard.

Description of function

The calculator offers the following functions:

- Basic mathematical operations
- Basic trigonometric functions
- Square root
- Exponential calculation
- Reciprocal value

Calculator

You can switch between the radian **RAD** or degrees **DEG** modes.

You can copy the result to the clipboard as well as paste the last stored value from the clipboard to the calculator.

The calculator saves the last ten calculations in the history. You can use these saved results for further calculations. You can clear the history manually.

26.11.1 Opening and closing the calculator

To open the calculator:

- Select the **calculator** on the control bar
- The control opens the calculator.

To close the calculator:

- Select the **calculator** when the calculator is open
- The control closes the calculator.

26.11.2 Selecting a result from the history

To select a result from the history for further calculations:

- Select **History**
- The control opens the calculator's history.
- Select the desired result
- Select **History**
- The control closes the calculator's history.

26.11.3 Deleting the history

To delete the calculator's history:

- Select **History**
- The control opens the calculator's history.
- Select **Delete**
- The control deletes the calculator's history.

26.12 Cutting data calculator

Application

With the cutting data calculator you can calculate the spindle speed and the feed rate for a machining process. You can load the calculated values into an opened feed rate or spindle speed dialog box in the NC program.

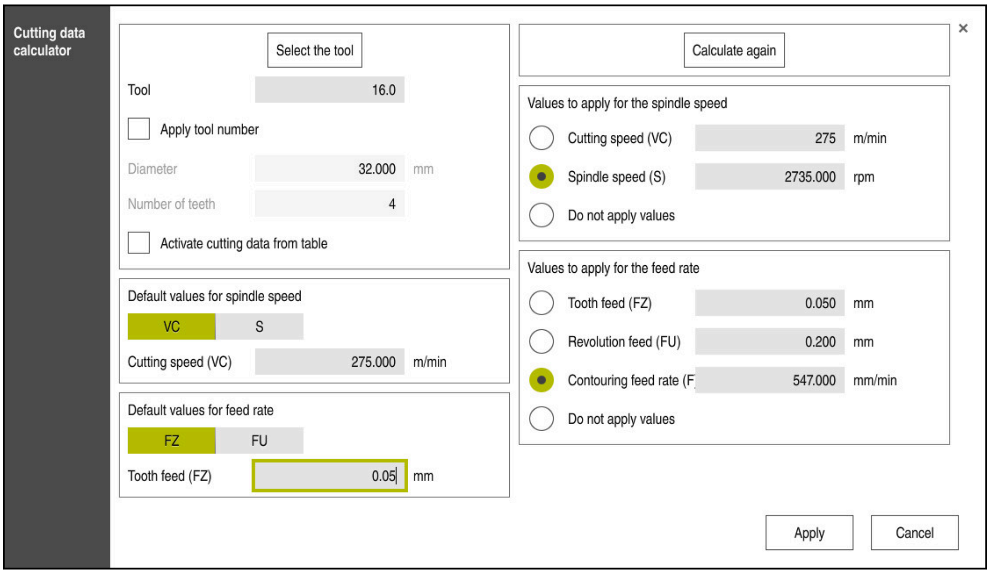
The control offers the **OCM cutting data calculator** for OCM cycles (option 167).

Further information: "OCM Cutting data calculator (option 167)", Page 604

Requirement

- Milling operation **FUNCTION MODE MILL**

Description of function



Window for the **Cutting data calculator**

On the left side of the cutting data calculator you enter the information. On the right side the control displays the calculated results.

If you select a tool defined in the tool management, the control automatically applies the tool diameter and number of teeth. If you activate the **Apply tool number** checkbox, the tool number in the current NC block is overwritten.

You can calculate the spindle speed as follows:

- Cutting speed **VC** in m/min
- Spindle speed **S** in rpm

You can calculate the feed rate as follows:

- Feed per tooth **FZ** in mm
- Feed per revolution **FU** in mm

Or you can use tables to calculate the cutting data.

Further information: "Calculation with tables", Page 1399

Applying values

After the cutting data have been calculated, you can specify which values the control should apply.

You can choose among the following for the spindle speed:

- **Cutting speed (VC)**
- **Spindle speed (S)**
- **Do not apply values**

You can choose among the following for the feed rate:

- **Tooth feed (FZ)**
- **Revolution feed (FU)**
- **Contouring feed rate (F)**
- **Do not apply values**

Calculation with tables

You must define the following in order to calculate the cutting data with tables:

- Workpiece material in the table **WMAT.tab**

  Further information: "Table for workpiece materials WMAT.tab", Page 1861
- Tool cutting material in table **TMAT.tab**

  Further information: "Table for tool materials TMAT.tab", Page 1861
- Combination of workpiece material and cutting material in the cutting data table ***.cut** or in the diameter-dependent cutting data table ***.cutd**

> Using the simplified cutting data table, you can determine speeds and feed rates using cutting data that are independent of the tool radius, e.g. **VC** and **FZ**.
>
> **Further information:** "Cutting data table *.cut", Page 1862
>
> If you require specific cutting data depending on the tool radius for your calculations, use the diameter-dependent cutting data table.
>
> **Further information:** "Diameter-dependent cutting data table *.cutd", Page 1863

- Parameters of the tool in tool management:
  - **R**: Tool radius
  - **LCUTS**: Number of cutting edges
  - **TMAT**: Cutting material from **TMAT.tab**
  - **CUTDATA**: Table row from the ***.cut** or ***.cutd** cutting data table

26.12.1 Opening the cutting data calculator

To open the cutting data calculator:

- Edit the desired NC block
- Select the syntax element for the feed rate or spindle speed
- Select **Cutting data calculator**
- > The control opens the **Cutting data calculator** window.

26.12.2 Calculating the cutting data with tables

The following prerequisites must be fulfilled in order to calculate the cutting data with tables:

- The **WMAT.tab** table exists
- The **TMAT.tab** table exists
- The ***.cut** or ***.cutd** table exists
- Tool material and cutting data table are assigned in the tool management

To calculate the cutting data with tables:

- Edit the desired NC block
- Open the **Cutting data calculator**
- Select **Activate cutting data from table**
- Use **Select material** to choose the workpiece material
- Use **Select type of machining** to choose the combination of workpiece material and tool material
- Select the desired values to be applied

Apply

- Press **Apply**
- > The control applies the calculated values in the NC block.

Notes

You cannot calculate the cutting data in turning mode (option 50) because the feed rate and spindle speed data are different in turning mode from milling mode.

In turning operations the feed rates are often defined in millimeters per revolution (mm/1) (**M136**), whereas the cutting data calculator always calculates feed rates in millimeters per minute (mm/min). Furthermore, the radius in the cutting data calculator is referenced to the tool; turning operations, however, require the workpiece diameter.

26.13 Message menu on the information bar

Application

In the message menu of the information bar, the control shows pending errors and notes. When opened, the control displays detailed information about the messages.

Description of function

The control uses the following symbols to differentiate between the types of messages:

Symbol	Message type	Meaning
	Error Question type	The control displays a dialog with several options you can select from. You cannot delete this error message: you can only choose one of the possible responses. If necessary, the control continues the dialog until the cause or correction of the error has been clearly determined.
	Reset error	The control must be restarted. This message cannot be cleared.
	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	Warning	You can continue without clearing the message. Most warnings can be cleared at any time; in some cases, the cause has to be eliminated first.
	Information	You can continue without clearing the message. You can clear the information at any time.
	Note	You can continue without clearing the message. The control displays the note until you press the next valid key.
		No pending messages

The message menu is collapsed by default.

The control displays messages upon various events, for example:

- Logical errors in the NC program
- Impossible contour elements
- Improper touch-probe inserts
- Hardware updates

Content

Collapsed message menu on the information bar

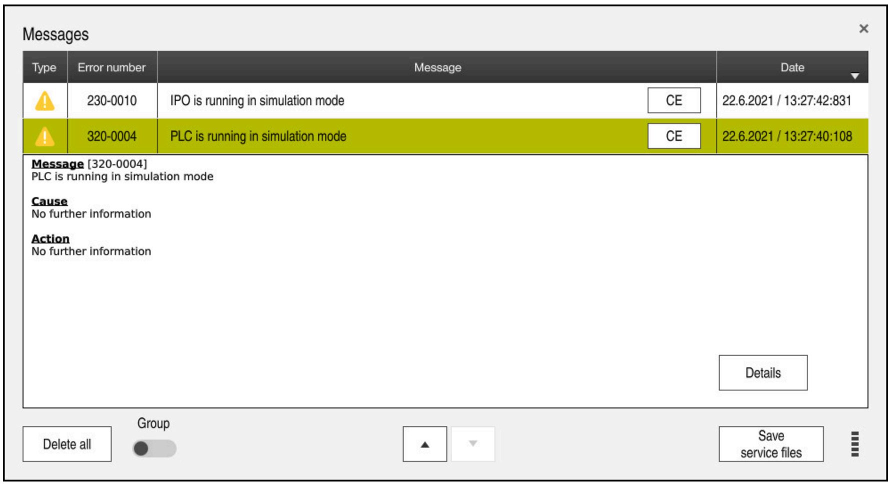
When the control displays a new message, the arrow to the left of the message blinks. Click or tap this arrow to confirm acknowledgment of the message; the control then minimizes the message.

The control displays the following information in the collapsed message menu:

- Message type
- Message
- Quantity of pending errors, warnings, and informational messages

Detailed messages

If you tap or click the symbol or within the message, the control expands the message menu.



Expanded message menu with pending messages

The control displays all pending messages in chronological order.

The message menu shows the following information:

- Message type
- Error number
- Message
- Date
- Additional information (cause, action)

Deleting messages

Messages can be deleted in the following ways:

- **CE** key
- **CE** button in the message menu
- **Delete all** button in the message menu

Details

Press the **Details** button to show or hide internal information about the message. This information is of importance in case servicing is necessary.

Group

If you activate the **Group** switch, the control displays all messages with the same error number in one row. This makes the list of messages shorter and easier to read.

Under the error number, the control displays the quantity of messages. Use **CE** to delete all messages of a group.

Service file

Use the **Save service files** button to create a service file.

A service file can be of use to the service technician during troubleshooting. The control saves data that provide information about the current machine and operation status, such as active NC programs up to 10 MB, tool data, and keystroke logs.

26.13.1 Creating a service file

To create a service file:

- Expand the message menu

Save service files

- Select **Save service files**
- > The control opens the **Save service file** window.
- Enter the file name

OK

- Press **OK**
- > The control saves the service file in the **TNC:\service** directory.

27

Simulation Workspace

27.1 Fundamentals

Application

In the **Editor** operating mode, you can use the **Simulation** workspace to graphically test whether NC programs are programmed correctly and run without collisions.

In the **Manual** and **Program Run** operating modes, the control shows the current traverse motions of the machine in the **Simulation** workspace.

Requirements

- Tool definitions according to the tool data from the machine
- Workpiece blank definition that is valid for a test run

  Further information: "Defining a workpiece blank with BLK FORM", Page 234

Description of function

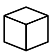
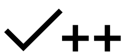
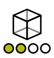
In the **Editor** operating mode, the **Simulation** workspace can be open for only one NC program at a time. If you want to open the workspace on a different tab, the control prompts you for confirmation.

The functions available in the simulation depend on the following settings:

- Selected type of model, for example **2.5D**
- Selected quality of model, for example **Medium**
- Selected mode, e.g. **Machine**

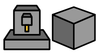
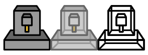
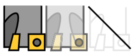
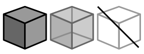
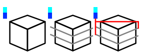
Icons in the Simulation workspace

The following symbols are shown in the **Simulation** workspace:

Symbol	Function
	**Visualization options** **Further information:** "Visualization options column", Page 1407
	**Workpiece options** **Further information:** "Workpiece options column", Page 1409
	Pre-defined views **Further information:** "Pre-defined views", Page 1414
	Export simulated workpiece as STL file **Further information:** "Exporting a simulated workpiece as STL file", Page 1415
	**Simulation settings** **Further information:** "Simulation settings window", Page 1411
 	Status of dynamic collision monitoring (DCM) in the simulation **Further information:** "Visualization options column", Page 1407
	Status of the **Advanced checks** function **Further information:** "Visualization options column", Page 1407
	Selected quality of model **Further information:** "Simulation settings window", Page 1411
T 3	Number of the active tool
00:00:00	Current program run-time

Visualization options column

In the **Visualization options** column you can define the following display modes and functions:

Symbol or switch	Function	Requirements
	Select **Machine** or **Workpiece** mode If you select the **Machine** mode, the control displays the defined workpiece, the collision objects, and the tool. In the **Workpiece** mode the control shows the workpiece to be simulated. Depending on the selected mode, different functions are available.	
Workpiece position	Use this function to define the position of the workpiece preset for the simulation. You can use a button to apply the current workpiece preset from the preset table. **Further information:** "Preset management", Page 949	■ **Machine** mode ■ Type of model: **2.5D**
	You can select between the following display modes for the machine: ■ **Original**: Shaded, opaque representation ■ **Semitransparent**: Transparent representation ■ **Wire-frame model**: Representation of the machine contours	■ **Workpiece** mode ■ Type of model: **2.5D**
	You can select between the following display modes for the tool: ■ **Original**: Shaded, opaque representation ■ **Semitransparent**: transparent representation ■ **Invisible**: The object is hidden	■ **Workpiece** mode ■ Type of model: **2.5D**
	You can select between the following display modes for the workpiece: ■ **Original**: Shaded, opaque representation ■ **Semitransparent**: transparent representation ■ **Invisible**: the object is hidden	■ **Workpiece** mode ■ Type of model: **2.5D**
	You can show the tool paths during the simulation. The control displays the center-line path of the tools. You can choose between the following display modes for the tool paths: ■ **None**: Do not show tool paths ■ **Feed**: Show tool paths with programmed feed rate ■ **Feedrate + FMAX**: Show tool paths with programmed feed rate and with programmed rapid traverse	■ **Workpiece** mode ■ Operating mode: **Editor**
DCM	Use this switch to activate or deactivate collision monitoring (DCM, option 40) for the simulation. **Further information:** "Dynamic Collision Monitoring (DCM) in the Editor operating mode", Page 1087	■ **Workpiece** mode ■ Operating mode: **Editor** ■ Type of model: **2.5D**
Advanced checks	Use this switch to activate the **Advanced checks** function. **Further information:** "Advanced checks in the simulation", Page 1109	■ Operating mode: **Editor**

Symbol or switch	Function	Requirements
Skip /	If an NC block is preceded by a **/** character, then the NC block is hidden. If you activate the **Skip /** switch, the control skips all hidden NC blocks in the simulation. **Further information:** "Hiding NC blocks", Page 1385	■ Operating mode: **Editor**
Pause at M1	If you activate this switch, the control pauses the simulation at each **M1** miscellaneous function in the NC program. **Further information:** "Overview of miscellaneous functions", Page 1225	■ Operating mode: **Editor**

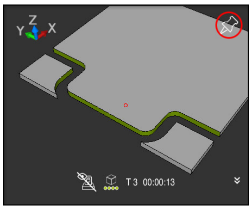
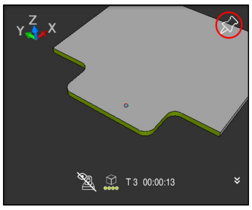
Workpiece options column

In the **Workpiece options** column, you can define the following simulation functions for the workpiece:

Switch or button	Function	Requirements
Measuring	Use this function to measure any points on the simulated workpiece. **Further information:** "Measuring function", Page 1417	■ **Workpiece** mode ■ Operating mode: **Editor** ■ Type of model: **2.5D**
Cutout view	Use this function to cut through the simulated workpiece along a plane. **Further information:** "Cutout view in the simulation", Page 1419	■ **Workpiece** mode ■ Operating mode: **Editor** ■ Type of model: **2.5D**
Highlight workpiece edges	Use this function to highlight the edges of the simulated workpiece.	■ **Workpiece** mode ■ Type of model: **2.5D**
Workpiece blank frame	The control uses this function to show the outside lines of the workpiece blank.	■ **Workpiece** mode ■ Operating mode: **Editor** ■ Type of model: **2.5D**
Finished part	Use this function to show a finished part that was defined with the help of the **BLK FORM FILE** function. **Further information:** "Cutout view in the simulation", Page 1419	■ **Workpiece** mode ■ Operating mode: **Editor** ■ Type of model: **2.5D**
Software limit switches	Use this function to activate the software limit switches of the machine for the active traverse range in the simulation. By simulating the limit switches you can check whether the working space of the machine is sufficient for the simulated workpiece. **Further information:** "Simulation settings window", Page 1411	■ Operating mode: **Editor**

Switch or button	Function	Requirements

Workpiece coloring	■ **Grayscale**
	The control displays the workpiece in various shades of gray.
	■ **Tool based**
	The control displays the workpiece in color. Each cutting tool is assigned a separate color.
	■ **Model comparison**
	The control displays a comparison between the workpiece blank and the finished part.
	Further information: "Model comparison", Page 1420
	■ **Monitoring**
	The control displays a heat map on the workpiece:
	  ■ Component heat map with **MONITORING HEAT MAP**
	  Further information: "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1146
	  Further information: "Cycles for monitoring", Page 1148
	  ■ Process heat map with **SECTION MONITORING**
	  Further information: "Process Monitoring (option 168)", Page 1152
		■ Type of model: **2.5D**
		■ **Model comparison** function only in **Workpiece** mode
		■ **Monitoring** function only in the **Program Run** operating mode

Reset the workpiece	Use this function to reset the workpiece back to the workpiece blank	■ Operating mode: **Editor** ■ Type of model: **2.5D**
Reset the tool paths	Use this function to reset the simulated tool paths.	■ **Workpiece** mode ■ Operating mode: **Editor**
Remove the chips	Use this function to remove from the simulation those parts of the workpiece that were cut off during machining.  Workpiece before clean-up  Workpiece after clean-up	■ Operating mode: **Editor** ■ Type of model: **3D**

Simulation settings window

The **Simulation settings** window is available only in the **Editor** operating mode.

The **Simulation settings** window consists of the following areas:

<table>
<tr><th>Area</th><th>Function</th></tr>
<tr><td>General</td><td>

Model type

None: fast line graphics without volume model
2.5D: quick 3D representation without undercuts
3D: realistic 3D representation with undercuts

Quality

Low: low-quality model, low memory use
Medium: normal-quality model, average memory use
High: high-quality model, uses much memory
Highest: best-quality model, uses very much memory

Mode

Milling
Turning
Grinding

Active kinemat.
Select the kinematics model for the simulation from a selection menu. The machine manufacturer enables the kinematics models.
Generate tool-usage file

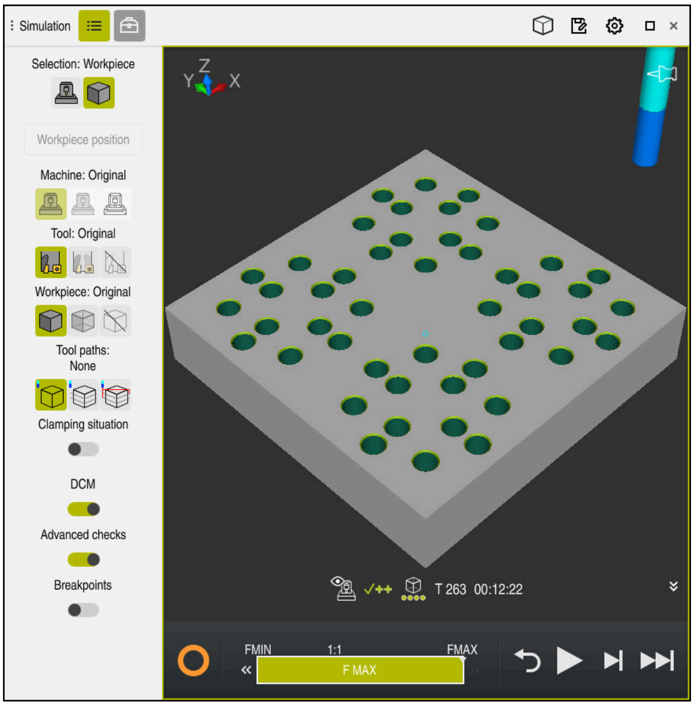
Never
Do not generate a tool-usage file
Once
Generate a tool-usage file for the next simulated NC program
Always
Generate a tool-usage file for every simulated NC program

Further information: "Channel settings", Page 1915</td></tr>
<tr><td>Traverse ranges</td><td>

Traverse ranges
In this selection menu you can choose one of the traverse ranges defined by the machine manufacturer, such as Limit1. In each traverse range the machine manufacturer defines different software limit switches for each axis of the machine. For example, the machine manufacturer defines traverse ranges for large machines with two separate working spaces.
Further information: "Workpiece options column", Page 1409
Active traverse ranges
This function shows the active traverse range and the values defined for within that range.
</td></tr>
</table>

Area	Function
Tables	You can select tables specifically for the **Editor** operating mode. The control uses the selected tables for the simulation. The selected tables are independent of any tables that are active in other operating modes. You use a selection menu to choose the tables. You can select the following tables for the **Simulation** workspace: ■ Tool table ■ Turning tool table ■ Datum table ■ Preset table ■ Grinding tool table ■ Dressing tool table **Further information:** "Tool Tables", Page 1813

Action bar



Simulation workspace in the **Editor** operating mode

In the **Editor** operating mode you can test NC programs by simulating them. The simulation helps to detect programming errors or collisions and to check the machining result visually.

The control shows the active tool and the machining time above the action bar.

The action bar contains the following symbols:

Symbol	Function
	Control-in-operation: The control uses the **Control-in-operation** symbol to show the current simulation status in the action bar and on the tab of the NC program: ■ White: no movement command ■ Green: active machining, axes are moving ■ Orange: NC program interrupted ■ Red: NC program stopped
	Simulation speed **Further information:** "Simulation speed", Page 1422
	Reset Return to the beginning of the program, reset transformations and the machining time
	Start
	Start in Single Block mode
	Run the simulation up to a certain NC block **Further information:** "Simulating an NC program up to a certain NC block", Page 1423

Simulation of tools

The control visualizes the following entries of the tool table in the simulation:

- **L**
- **LCUTS**
- **LU**
- **RN**
- **T-ANGLE**
- **R**
- **R2**
- **KINEMATIC**
- Delta values from the tool table

  Delta values from the tool table increase or decrease the size of the simulated tool. Delta values from the tool call shift the tool in the simulation.

  Further information: "Tool compensation for tool length and radius", Page 1036

  Further information: "Tool table tool.t", Page 1814

The control visualizes the following entries of the turning tool table in the simulation:

- **ZL**
- **XL**
- **YL**
- **RS**
- **T-ANGLE**
- **P-ANGLE**
- **CUTLENGTH**
- **CUTWIDTH**

If the **ZL** and **XL** columns are defined in the turning tool table, the indexable insert is displayed and the base body is shown schematically.

Further information: "Turning tool table toolturn.trn (option 50)", Page 1823

The control visualizes the following entries of the grinding tool table in the simulation:

- **R-OVR**
- **LO**
- **B**
- **R_SHAFT**

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 1828

The control displays the tool in the following colors:

- Turquoise: tool length
- Red: length of cutting edge and tool is engaged
- Blue: length of cutting edge and tool is retracted

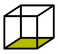
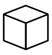
27.2 Pre-defined views

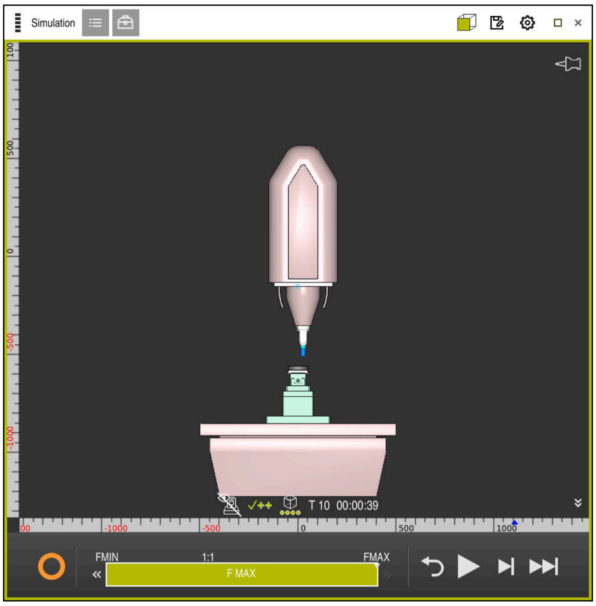
Application

In the **Simulation** workspace, you can choose between various pre-defined views in order to align the workpiece. This allows you to position the workpiece more quickly for the simulation.

Description of function

The control provides the following pre-defined views:

Symbol	Function
	Plan view
	Bottom view
	Front view
	Back view
	Side view (left side)
	Side view (right side)
	Isometric view



Front view of the simulated workpiece in **Machine** mode

27.3 Exporting a simulated workpiece as STL file

Application

In the simulation, you can use the **Save** function to save the current status of the simulated workpiece as a 3D model in STL format.

The file size of the 3D model depends on the complexity of the geometry and the selected model quality.

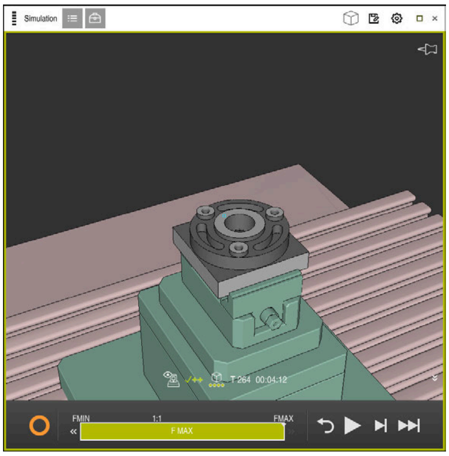
Related topics

- Using an STL file as workpiece blank

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 239
- Modifying an STL file in the **CAD-Viewer** (option 152)

  Further information: "Generating STL files with 3D mesh (option 152)", Page 1372

Description of function



Simulated workpiece

This function can be used only in the **Editor** mode.

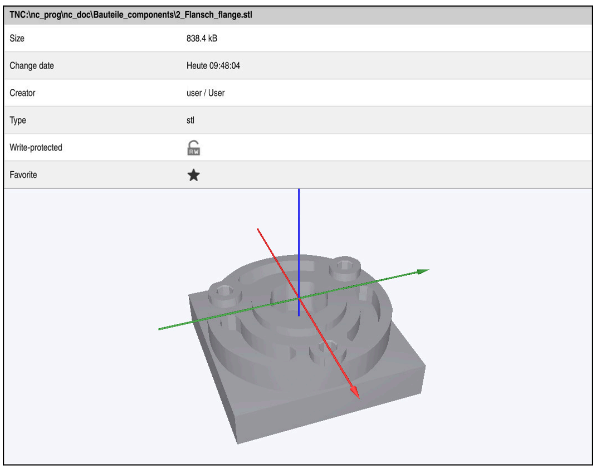
The control can only display STL files with up to 30,000 triangles. If the exported 3D model has too many triangles, due to the excessively high model quality, then you cannot use the exported 3D model on the control.

In this case, reduce the model quality in the simulation.

Further information: "Simulation settings window", Page 1411

You can also use the **3D mesh** function to reduce the number of triangles (option 152).

Further information: "Generating STL files with 3D mesh (option 152)", Page 1372



Simulated workpiece as saved STL file

27.3.1 Saving a simulated workpiece as STL file

To save a simulated workpiece as an STL file:

- ▶ Simulate workpiece
- ▶ Select **Save**
- > The control opens the **Save as** window.
- ▶ Enter the desired file name
- ▶ Select **Create**
- > The control saves the created STL file.

27.4 Measuring function

Application

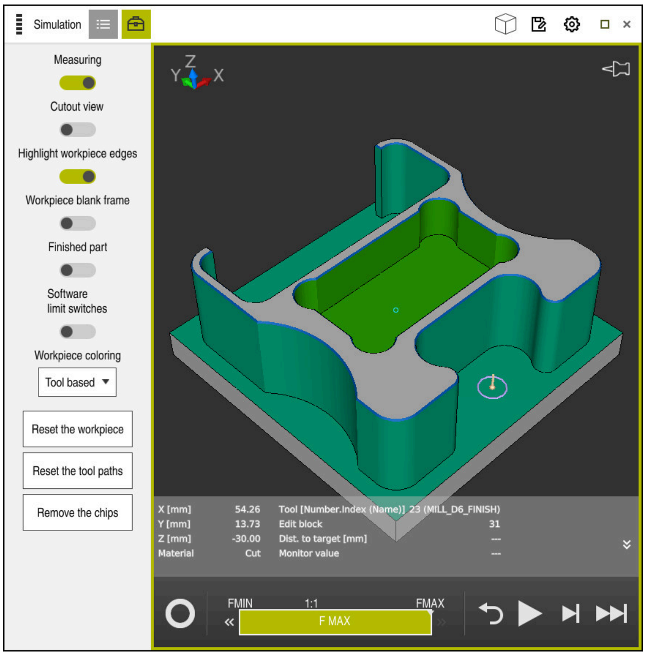
Use the measuring function to measure any points on the simulated workpiece. The control shows various pieces of information about the measured surface.

Requirement

- **Workpiece** mode

Description of function

If you measure a point on the simulated workpiece, the cursor always locks onto the currently selected surface.



Measured point on simulated workpiece

The control shows the following information about the measured surface:

- Measured positions in the **X**, **Y**, and **Z** axes
- Status of the machined surface
  - **Material Cut** = Surface that has been machined
  - **Material NoCut** = Surface that has not been machined
- Cutting tool
- NC block currently running in the NC program
- Distance between the measured surface and the finished part
- Relevant values of monitored machine components (option 155)

  Further information: "Component Monitoring with MONITORING HEATMAP (option 155)", Page 1146

27.4.1 Measuring the difference between the workpiece blank and the finished part

To measure the difference between the workpiece blank and the finished part:

- Select an operating mode, e.g. **Editor**
- Open an NC program with a workpiece blank and finished part defined in **BLK FORM FILE**
- Open the **Simulation** workspace
- Select the **Tool options** column
- Activate the **Measuring** switch
- Select the **Workpiece coloring** selection menu

- Select **Model comparison**
- > The control displays the workpiece blank and finished part defined in the **BLK FORM FILE** function.
- Start the simulation
- > The control simulates the workpiece.
- Select the desired point on the simulated workpiece
- > The control displays the difference in the dimension between the simulated workpiece and the finished part.

> The control uses the **Model comparison** function to identify dimensional differences between the simulated workpiece and the finished part first in color, starting with differences greater than 0.2 mm.

Notes

- If you need to compensate for tools, you can use the measuring function to determine the tool to be compensated for.
- If you notice an error in the simulated workpiece, you can use the measuring function to determine the NC block that causes the error.

27.5 Cutout view in the simulation

Application

In the Cutout view you can cut through the simulated workpiece along any axis. This enables you to check holes and undercuts in the simulation, for example.

Requirement

- **Workpiece** mode

Description of function

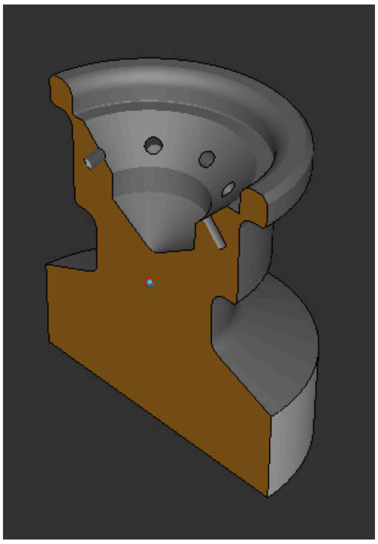
The Cutout view can be used only in the **Editor** mode.

The position of the sectional plane is shown as a percent value when it is shifted in the simulation. The sectional plane is retained until the control is restarted.

27.5.1 Shifting the sectional plane

To shift the sectional plane:

- Select the **Editor** operating mode
- Open the **Simulation** workspace
- Select the **Visualization options** column
- Select **Workpiece** mode
- > The control shows the workpiece view.
- Select **Workpiece options**
- Activate the **Cutout view** switch
- > The control activates the **Cutout view**.
- Use the selection menu to choose the desired sectional axis, such as the Z axis
- Use the slider to specify the desired percent value
- > The control simulates the workpiece with the selected sectional settings.



Simulated workpiece in the **Cutout view**

27.6 Model comparison

Application

With the **Model comparison** function you can compare the blank and finished part in STL or M3D format with each other.

Related topics

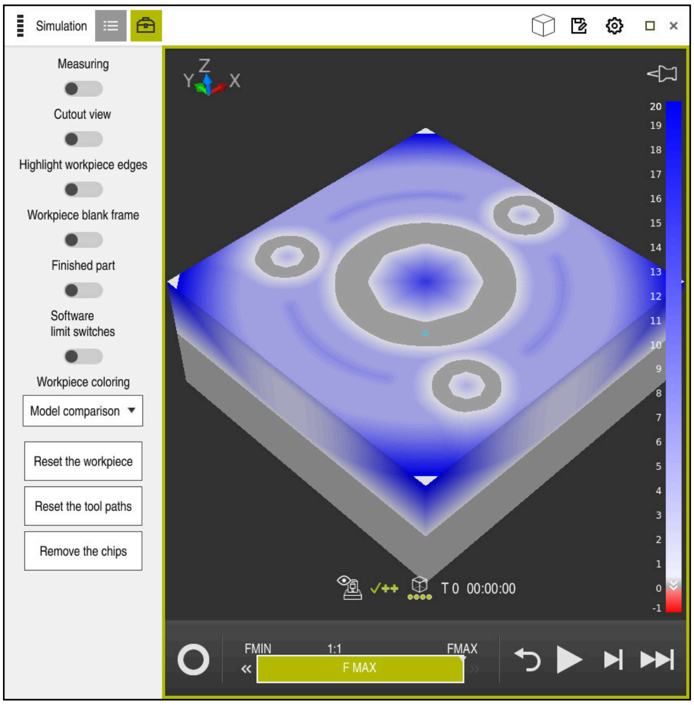
- Programming the blank and finished part with STL files

  Further information: "STL file as workpiece blank with BLK FORM FILE", Page 239

Requirements

- STL file or M3D file of workpiece blank and finished part
- **Workpiece** mode
- Workpiece blank definition with **BLK FORM FILE**

Description of function



The control uses the **Model comparison** function to show the difference in material between the models being compared. The control uses a color transition from white to blue to show the difference in material. The more material there is covering the finished part model, the deeper the blue is. When material is removed from the finished part model, the control displays this removal in red.

Notes

- The control uses the **Model comparison** function to identify dimensional differences between the simulated workpiece and the finished part, starting with differences greater than 0.2 mm.
- Use the measuring function to measure the exact dimensional difference between the workpiece blank and the finished part.

  Further information: "Measuring the difference between the workpiece blank and the finished part", Page 1419

27.7 Center of rotation in the simulation

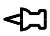
Application

By default, the center of rotation in the simulation is at the center of the model. When you zoom in, the center of rotation is always shifted to the center of the model. If you want to rotate the simulation around a specific point, then you can define the center of rotation manually.

Description of function

Use the **Center of rotation** function to manually set the center of rotation for the simulation.

The control shows the **Center of rotation** symbol as follows, depending on the status:

Symbol	Function
	The center of rotation is at the center of the model.
	The symbol blinks. The center of rotation can be shifted.
	The center of rotation was set manually.

27.7.1 Setting the center of rotation to a corner of the simulated workpiece

To set the center of rotation to a corner of the workpiece:

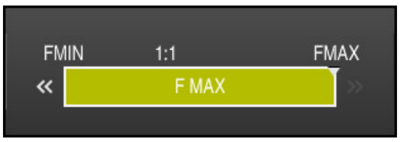
- Select an operating mode, e.g. **Editor**
- Open the **Simulation** workspace
- > The center of rotation is at the center of the model.

- Select **Center of rotation**
- > The control switches the **Center of rotation** symbol. The symbol blinks.
- Select a corner of the simulated workpiece
- > The center of rotation is defined. The control switches the **Center of rotation** symbol to "set".

27.8 Simulation speed

Application

You can use a slider to select any speed for the simulation.



Description of function

This function can be used only in the **Editor** operating mode.

The standard speed for the simulation is set to **FMAX**. If you change the simulation speed, then this change is retained until the control is restarted.

You can change simulation speed before as well as during the simulation.

The control provides the following options:

Button	Functions
FMIN	Activate minimum feed rate (**0.01*T**)
«	Reduce the feed rate
1:1	Feed-rate at 1:1 (real-time)
»	Increase the feed rate
FMAX	Activate maximum feed rate (**FMAX**)

27.9 Simulating an NC program up to a certain NC block

Application

If you want to check a critical point in the NC program then you can simulate the NC program up to a specific NC block that you specify. Once the NC block is reached in the simulation, the control stops the simulation automatically. Starting from this NC block you can then continue the simulation, for example in **Single Block** mode or at a lower simulation speed.

Related topics

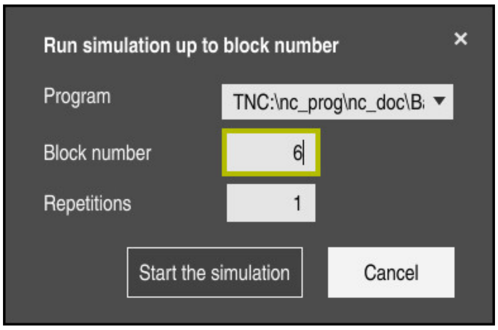
- Possibilities in the action bar

  Further information: "Action bar", Page 1413
- Simulation speed

  Further information: "Simulation speed", Page 1422

Description of function

This function can be used only in the **Editor** operating mode.



The **Run simulation up to block number** window with a defined NC block

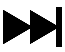
The following settings options are offered in the **Run simulation up to block number** window:

- **Program**
  This field offers a selection menu in which you can choose to simulate up to a specific NC block in the active main program or in a called program.
- **Block number**
  In the **Block number** field, you enter the number of the NC block up to which the simulation should run. The number of the NC block refers to the NC program selected in the **Program** field.
- **Repetitions**
  Use this field if the desired NC block is located within a program-section repeat. Enter in this field up to which iteration of the program-section repeat the simulation should run.
  If you enter **1** or **0** in the **Repetitions** field, the control simulates up to the first iteration of the program section (repetition "0").
  Further information: "Program-section repeats", Page 335

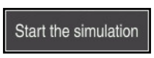
27.9.1 Simulating an NC program up to a certain NC block

To simulate up to a specific NC block:

- ▶ Open the **Simulation** workspace



- ▶ Select **Run simulation up to block number**
- > The control opens the **Run simulation up to block number** window.
- ▶ Use the selection menu in the **Program** field to specify the main program or called program
- ▶ Enter the number of the desired NC block in the **Block number** field
- ▶ If the block involves a program-section repeat, enter the number of the iteration of the program-section repeat in the **Repetitions** field



- ▶ Select **Start the simulation**
- > The control simulates the workpiece up to the selected NC block.

28

Touch Probe Functions in the Manual Operating Mode

28.1 Fundamentals

Application

The touch probe functions allow you to set presets on the workpiece, measure the workpiece, and determine and compensate for workpiece misalignment.

Related topics

- Automatic touch probe cycles
  Further information: "Programmable Touch Probe Cycles", Page 1449
- Preset table
  Further information: "Preset table", Page 1851
- Datum table
  Further information: "Datum table", Page 1858
- Reference systems
  Further information: "Reference systems", Page 934
- Preassigned variables
  Further information: "Preassigned Q parameters", Page 1273

Requirements

- Calibrated workpiece touch probe
  Further information: "Calibrating the workpiece touch probe", Page 1439

Description of function

The control provides the following functions for setting up the machine in the **Manual** operating mode in the **Setup** application:

- Define the workpiece preset
- Determine and compensate for workpiece misalignment
- Calibrate the workpiece touch probe
- Calibrate the tool touch probe
- Measure the tool

Within the functions, the control provides the following probing methods:

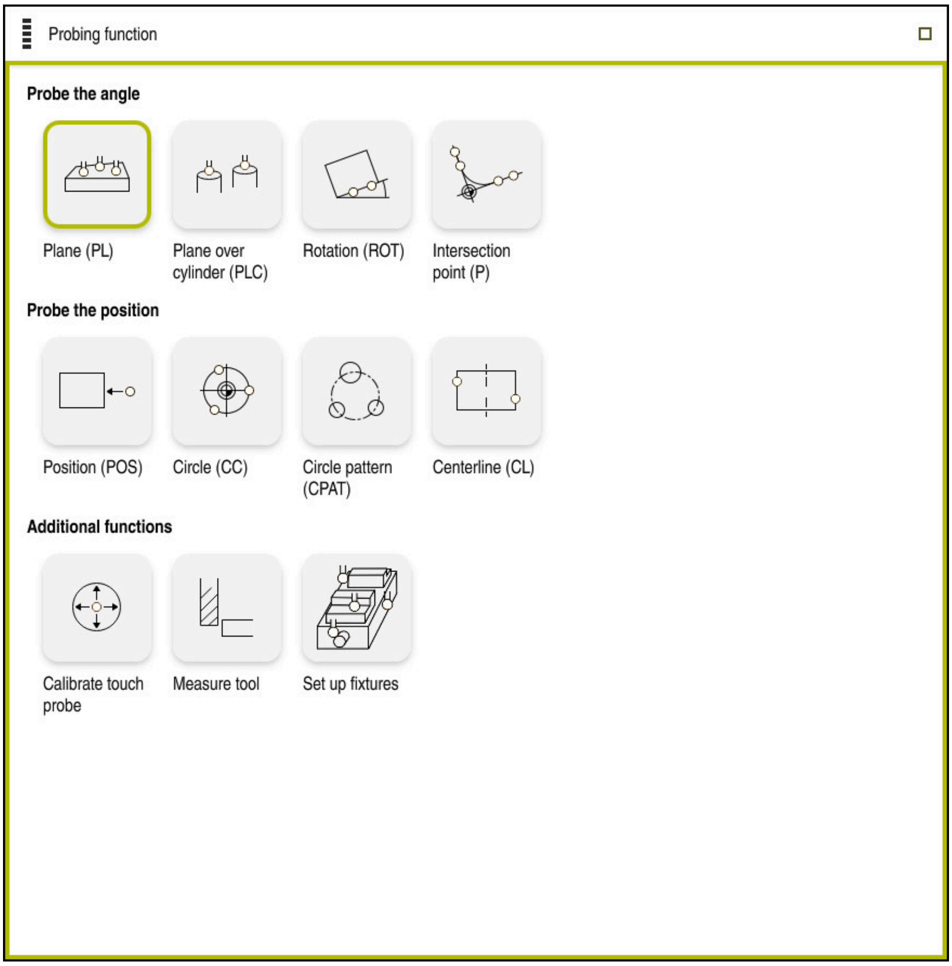
- Manual probing method

  You position and start individual probing processes manually within a touch probe function.

  Further information: "Setting a preset in a linear axis", Page 1433
- Automatic probing method

  You manually position the touch probe to the first probing point before the start of the probing routine and fill out a form with the individual parameters for the respective touch probe function. When you start the touch probe function, the control automatically positions and automatically performs probing.

  Further information: "Determining the circle center point of a stud using the automatic probing method ", Page 1435



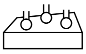
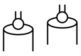
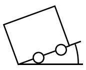
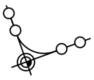
Probing function workspace

Overview

The touch probe functions are structured in the following groups:

Probe the angle

The **Probe the angle** group contains the following touch probe functions:

Button	Function
Plane (PL) 	Use the **Plane (PL)** function to determine the solid angle of a plane. You then save the values in the preset table or align the plane.
Plane over cylinder (PLC) 	Use the **Plane over cylinder (PLC)** function to probe one or two cylinders, each at two different heights. The control calculates the solid angle of a plane from the points probed. You then save the values in the preset table or align the plane.
Rotation (ROT) 	Use the **Rotation (ROT)** function to determine the skew of a workpiece using a straight line. Then save the determined skew as a basic transformation or offset in the preset table. **Further information:** "Determining and compensating the rotation of a workpiece", Page 1436
Intersection point (P) 	Use the **Intersection point (P)** function to probe four probing objects. The probing objects can be either positions or circles. The control determines the intersection of the axes and the skew of the workpiece from the objects that have been probed. You can set the intersection point as a preset. You can transfer the determined skew to the preset table as a basic transformation or as an offset.

The control interprets a basic transformation as a basic rotation, and an offset as a table rotation.

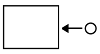
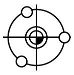
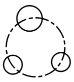
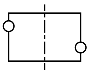
Further information: "Preset table", Page 1851

You can compensate for the workpiece misalignment by rotating the table only if the machine is designed with a rotary table axis that is oriented perpendicularly with respect to the workpiece coordinate system **W-CS**.

Further information: "Comparison of offset and 3D basic rotation", Page 1446

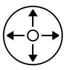
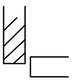
Probe the position

The **Probe the position** group contains the following touch probe functions:

Button	Function
Position (POS) 	You can use the **Position (POS)** function to probe a position in the X axis, Y axis or Z axis. **Further information:** "Setting a preset in a linear axis", Page 1433
Circle (CC) 	The **Circle (CC)** function is used to determine the coordinates of a circle center point, e.g. for a hole or for a stud. **Further information:** "Determining the circle center point of a stud using the automatic probing method ", Page 1435
Circle pattern (CPAT) 	The **Circle pattern (CPAT)** function is used to determine the center point coordinates of a circle pattern.
Centerline (CL) 	The **Centerline (CL)** function is used to determine the center point of a ridge or slot.

Additional functions group

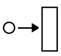
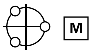
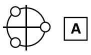
The **Additional functions** group contains the following touch probe functions:

Button	Function
Calibrate touch probe 	The **Calibrate touch probe** function is used to determine the length and radius of a workpiece touch probe. **Further information:** "Calibrating the workpiece touch probe", Page 1439
Measure tool 	The **Measure tool** function allows you to measure tools by scratching. In this function, the control supports milling tools, drilling tools and turning tools.
Set up fixtures 	The **Set up fixtures** function is used to determine the position of a clamping device in the machine space using a workpiece touch probe. **Further information:** "Spannmittel in Kollisionsüberwachung einmessen", Page

Buttons

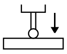
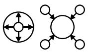
General buttons in the touch probe functions

The following buttons are available, depending on the selected touch probe function:

Button	Function
	Finish the active touch probe function
	Open the **Change the preset** window Select the workpiece preset in the **Change the preset** window and adapt arbitrary values to the preset table. **Further information:** "Preset table", Page 1851
	Display help graphics for the selected touch probe function
	Select the probing direction
	Apply the actual position
	Approaching and probing points on a straight surface manually
	Approaching and probing points on a stud or in a hole manually
	Approaching and probing points on a stud or in a hole automatically After the last touching process and if the opening angle contains the value 360°, the control positions the workpiece touch probe back to the position it had prior to starting the probing function.

Calibration buttons

The control offers the following ways for calibrating a 3D touch probe:

Button	Function
	Calibrating the length of a 3D touch probe
	Calibrating the radius of a 3D touch probe
Apply calibration data	Transferring values from the calibration process into tool management

Further information: "Calibrating the workpiece touch probe", Page 1439

You can calibrate a 3D touch probe by using a calibration standard, e.g. a calibrating ring.

The control provides the following options:

Button	Function
	Measure the radius and the center offset using a calibration ring
	Measure the radius and the center offset using a stud or a calibration pin
	Measure the radius and the center offset using a calibration sphere Optionally calibrating the tool touch probe in 3D (option 92) **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1062 **Further information:** "3D calibration (option 92)", Page 1440

Buttons in the Working plane is inconsistent! window

If the positions of the rotary axes do not match the tilting situation in the **3-D rotation** window, the control opens the **Working plane is inconsistent!** window.

The control offers the following functions in the **Working plane is inconsistent!** window:

Button	Function
3-D ROT Apply status	The **3-D ROT Apply status** function transfers the position of the rotary axes into the **3-D rotation** window. **Further information:** "3-D rotation window (option 8)", Page 1022
3-D ROT Ignore status	The **3-D ROT Ignore status** function makes the control calculate the probing results, assuming that the rotary axes are in their zero position.
Align the rotary axes	The **Align the rotary axes** function aligns the rotary axes to the active tilting situation in the **3-D rotation** window.

Buttons for measured values

After executing a touch probe function, you select the desired control reaction.

The control offers the following functions:

Button	Function
Compensate the active preset	The **Compensate the active preset** function transfers the measuring result into the active line of the preset table. **Further information:** "Preset table", Page 1851
Write the datum	The **Write the datum** function transfers the measuring result into a desired line of the datum table. **Further information:** "Datum table", Page 1858
Align rotary table	The **Align rotary table** function aligns the rotary axes mechanically according to the measuring result.

Log file of touch probe cycles

After executing the respective touch-probe cycle, the control writes the measured values to the TCHPRMAN.html file.

You can check the readings of past measurements in the **TCHPRMAN.html** file.

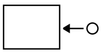
If you have not defined a path in the machine parameter **FN16DefaultPath** (no. 102202), the control will store the TCHPRMAN.html file directly under **TNC:**.

If you run several touch probes cycles in a row, the control stores the measured values below each other.

28.1.1 Setting a preset in a linear axis

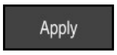
To probe the preset in any axis:

- Select the **Manual** operating mode
- Call the workpiece touch probe as a tool
- Select the **Setup** application



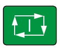
- Select the **Position (POS)** touch probe function
- > The control opens the **Position (POS)** touch probe function.

- Select **Change the preset**
- > The control opens the **Change the preset** window.
- Select the desired row of the preset table
- > The control highlights the selected line in green.

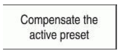


- Press **Apply**
- > The control activates the selected line as the workpiece preset.
- Using the axis keys, position the workpiece touch probe at the desired probing position, e.g. above the workpiece in the workspace

- Select the probing direction, e.g. **Z-**



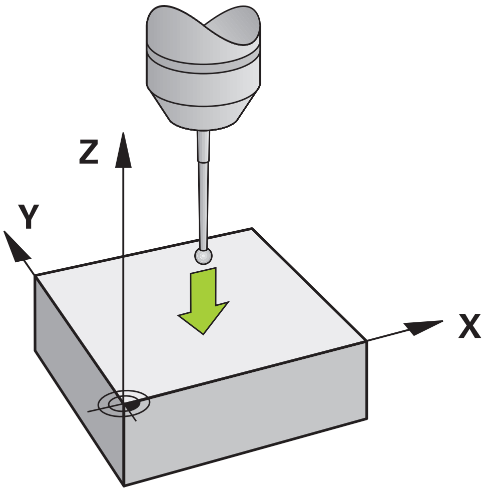
- Press the **NC start** key
- > The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
- > The control shows the measurement results.
- In the **Nominal value** area, enter the new preset of the probed axis, e.g. **1**



- Select **Compensate the active preset**
- > The control enters the defined nominal value in the preset table.

> When you have completed the probing process in the first axis, you can probe up to two additional axes using the **Position (POS)** probing function.

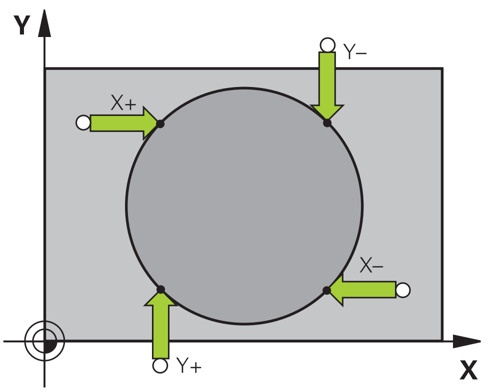
- Select **Exit probing**
- > The control closes the **Position (POS)** probing function.



28.1.2 Determining the circle center point of a stud using the automatic probing method

To probe a circle center point:

- Select the **Manual** operating mode
- Call the workpiece touch probe as a tool
  Further information: "Manual operation application", Page 180
- Select the **Setup** application
- Select **Circle (CC)**
- > The control opens the **Circle (CC)** probing function.
- If necessary, select another preset for the probing process
- Select measuring method **A**
- Select **Type of contour**, e.g. stud
- Enter **Diameter**, e.g. 60 mm
- Enter **Starting angle**, e.g. –180°
- Enter **Angular length**, e.g. 360°
- Position the 3D touch probe at the desired probing position next to the workpiece and below the workpiece surface
- Select the probing direction, e.g. **X+**
- Turn the feed rate potentiometer to zero
- Press the **NC start** key
- Slowly turn on the feed rate potentiometer
- > The control executes the touch probe function based on the data entered.
- > The control shows the measurement results.
- In the **Nominal value** area, enter the new preset of the scanned axes, e.g. **0**
- Select **Compensate the active preset**
- > The control sets the preset to the entered nominal value.
- Select **Exit probing**
- > The control closes the **Circle (CC)** probing function.



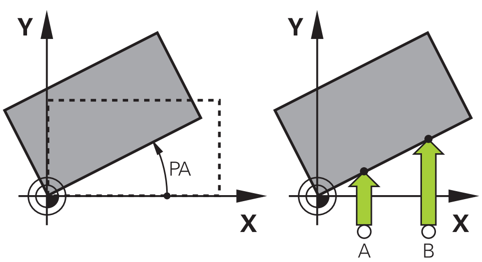
28.1.3 Determining and compensating the rotation of a workpiece

To probe the rotation of a workpiece:

- Select the **Manual** operating mode
- Call the 3D touch probe as a tool
- Select the **Setup** application
- Select **Rotation (ROT)**
- > The control opens the **Rotation (ROT)** probing function.
- If necessary, select another preset for the probing process
- Position the 3D touch probe at the desired probing position in the workspace
- Select the probing direction, e.g. **Y+**
- Press the **NC start** key
- > The control executes the first probing process and limits the subsequently selectable probing directions.
- Position the 3D touch probe at the second probing position in the workspace
- Press the **NC start** key
- > The control executes the probing process and then shows the measurement results.
- Select **Compensate the active preset**
- > The control transfers the determined basic rotation to the **SPC** column of the active line of the preset table.

> Depending on the tool axis, the measurement result can also be written to another column of the preset table, e.g. **SPA**.

- Select **Exit probing**
- > The control closes the **Rotation (ROT)** probing function.



28.1.4 Using touch probe functions with mechanical probes or dial gages

If your machine does not have an electronic 3D touch probe, you can use all manual touch probe functions with manual probing methods with mechanical buttons or with scratching.

For this, the control provides the **Accept position** button.

To determine a basic rotation with a mechanical probe:

- Select the **Manual** operating mode
- Replace tool, e.g. analog 3D probe or feeler lever gage
- Select the **Setup** application
- Select the **Rotation (ROT)** probing function
- Select the probing direction, e.g. **Y+**
- Move the mechanical probe to the first position to be captured by the control.
- Select **Accept position**
- > The control saves the current position.
- Move the mechanical probe to the next position to be captured by the control.
- Select **Accept position**
- > The control saves the current position.
- Select **Compensate the active preset**
- > The control transfers the determined basic rotation to the active line of the preset table.

> The determined angles have different effects depending on whether they are transferred as an offset or as a basic rotation to the corresponding table.
>
> **Further information:** "Comparison of offset and 3D basic rotation", Page 1446

- Select **Exit probing**
- > The control closes the **Rotation (ROT)** probing function.

Notes

- When you use a non-contacting tool touch probe, use touch probe functions by the third-party manufacturer, e.g. with a laser touch probe. Refer to your machine manual.
- The accessibility of the pallet preset table in the touch probe functions depends on the machine manufacturer's configuration. Refer to your machine manual.
- The use of touch probe functions deactivates the global program settings (GPS, option 44) temporarily.

  Further information: "Global Program Settings (GPS, option 44)", Page 1132
- You can use the manual touch probe functions only with restrictions in turning mode (option 50).
- You must calibrate the touch probe separately in turning mode. The factory default setting of the worktable may vary between milling mode and turning mode, which is why you must calibrate the touch probe without any center offset in turning mode. You can create a tool index for storing the additionally calibrated tool data in the same tool.

  Further information: "Indexed tool", Page 250
- When probing while the guard door is open and spindle orientation to probing direction is active, the number of spindle revolutions is limited. When the maximum permitted number of spindle revolutions is reached, the direction of spindle rotation changes and the control may no longer orient the spindle on the shortest path.
- If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine manufacturer has defined.
- When writing into an empty line of the preset table, the control automatically fills the other columns with values. To define a preset completely, you must determine the values in all axes and write them into the preset table.
- If no tool touch probe is inserted, the actual position can be captured with **NC START**. The control displays a warning that no probing movement is carried out in that case.
- Recalibrate the workpiece touch probe in the cases below:
  - Initial configuration
  - Broken stylus
  - Stylus replacement
  - Change in the probe feed rate
  - Irregularities caused, for example, when the machine heats up
  - Change of active tool axis

Definition

Spindle tracking

If the **Track** parameter in the touch probe table is active, the control orients the workpiece probing system so that the same position is always used for probing. By deflecting in the same direction, you can reduce the measurement error to the repeatability of the workpiece probing system. This behavior is called spindle tracking.

28.2 Calibrating the workpiece touch probe

Application

In order to precisely specify the actual trigger point of a 3D touch probe, you must first calibrate the touch probe, otherwise the control cannot provide precise measuring results.

During 3D calibration, you determine the angle-dependent deflection behavior of a workpiece touch probe in any probing direction (option 92).

Related topics

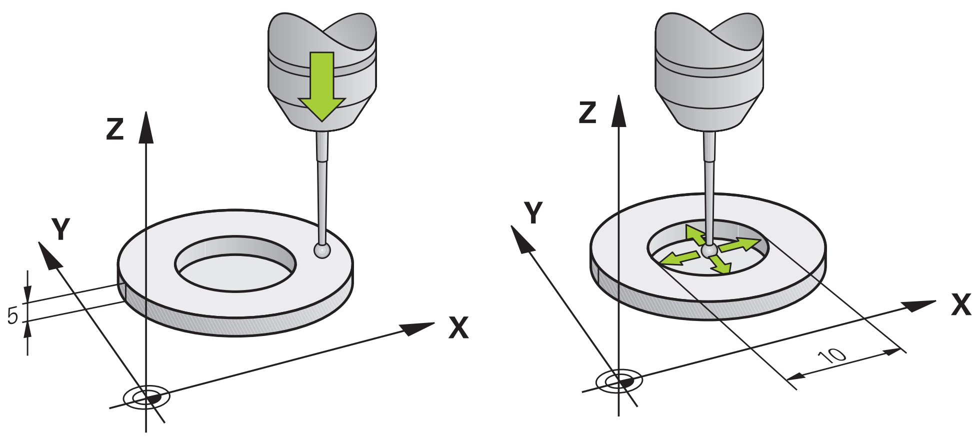
- Calibrate the workpiece touch probe automatically

  Further information: "Touch Probe Cycles: Calibration", Page 1679
- Touch probe table

  Further information: "Touch probe table tchprobe.tp", Page 1839
- 3D radius compensation depending on the contact angle (option 92)

  Further information: "3D radius compensation depending on the tool contact angle (option 92)", Page 1062

Description of function



During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The effective length of the workpiece touch probe refers to the tool carrier preset.

Further information: "Tool carrier reference point", Page 245

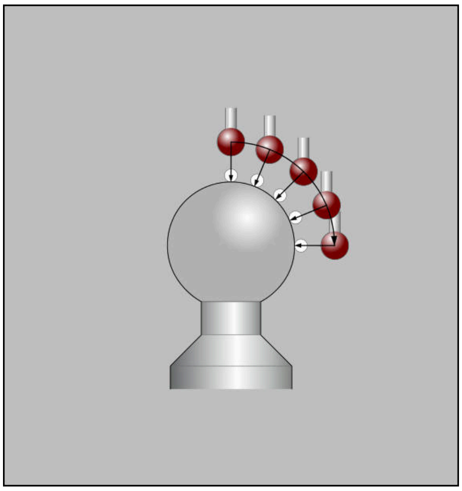
You can calibrate the workpiece touch probe with various tools. For example, the workpiece touch probe can be calibrated using an overmilled surface in length and a calibration ring in the radius. This creates a reference between the workpiece touch probe and the tools in the spindle. In this procedure, measured tools and the calibrated workpiece touch probe correspond using the tool presetting device.

3D calibration (option 92)

In addition to calibrating with a calibration sphere, the control also enables the touch probe to be calibrated dependent on the angle. For this purpose the control probes the calibration sphere in a quarter circle in the perpendicular. The 3D calibration data specifies the deflection behaviour of the touch probe in any probing direction.

The control saves the deviations in a compensation value table ***.3DTC** in the folder **TNC:\system\3D-ToolComp**.

The control creates a specific table for each calibrated touch probe. In the tool table the **DR2TABLE** column is automatically referenced to this.



3D calibration

Reversal measurement

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or stud (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

HEIDENHAIN touch probes are predefined as to whether or how a touch probe can be oriented. Other touch probes are configured by the machine manufacturer.

When calibrating the radius, up to three circular measurements can be taken depending on the possible orientation of the workpiece touch probe. The first two circular measurements determine the center offset of the workpiece touch probe. The third circular measurement determines the effective stylus tip radius. If orientation of the spindle is not possible or only a certain orientation is possible due to the workpiece touch probe, circular measurements are omitted.

28.2.1 Calibrating the length of the workpiece touch probe

To calibrate a workpiece touch probe using an overmilled surface in length:

- Measure the end milling cutter on the tool presetting device
- Store the measured end milling cutter in the tool magazine of the machine
- Enter the tool data of the end milling cutter in tool management
- Clamp the workpiece blank

- Select the **Manual** operating mode
- Replace the end milling cutter in the machine
- Switch on spindle, e.g. with **M3**
- Use the handwheel to scratch the workpiece blank

  Further information: "Setting a preset with milling cutters", Page 950
- Set preset in the tool axis, e.g. **Z**
- Position the end milling cutter next to the workpiece blank
- Set a small value in the tool axis, e.g. -0.5 mm
- Overmill the workpiece blank using the handwheel
- Set the preset again in the tool axis, e.g. **Z=0**
- Switch off spindle, e.g. with **M5**
- Replace the tool touch probe
- Select the **Setup** application
- Select **Calibrate touch probe**
- Select the **Length calibration** measurement method
- > The control displays the current calibration data.
- Enter the reference surface position, e.g. **0**
- Position the workpiece touch probe close to the surface of the overmilled area

> Check that the area to be probed is flat and free of chips before you start the touch probe function.

- Press the **NC start** key
- > The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
- Check results

Apply calibration data

- Select **Apply calibration data**
- > The control transfers the calibrated length of the 3D touch probe to the tool table.
- Select **Exit probing**
- > The control closes the **Calibrate touch probe** function.

28.2.2 Calibrating the radius of the workpiece touch probe

To calibrate a workpiece touch probe using a setting ring in the radius:

- Clamp the setting ring on the machine table, e.g. with clamps
- Select the **Manual** operating mode
- Position the 3D touch probe in the hole of the setting ring

> Make sure that the stylus tip is completely recessed into the calibration ring. This causes the control to probe with the largest point of the stylus tip.

- Select the **Setup** application
- Select **Calibrate touch probe**
- Select **Radius** measurement method
- Select **Setting ring** calibration standard
- Enter the diameter of the ring gauge
- Enter the start angle
- Enter the number of touch points
- Press the **NC Start** key
- \> The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- Check results
- Select **Apply calibration data**
- \> The control stores the calibrated radius of the 3D touch probe in the tool table.
- Select **Exit probing**
- \> The control closes the **Calibrate touch probe** function.

28.2.3 3D calibration of workpiece touch probe (option 92)

To calibrate a workpiece touch probe using a calibration sphere in the radius:

- Clamp the setting ring on the machine table, e.g. with clamps
- Select the **Manual** operating mode
- Position the workpiece touch probe centrally above the sphere
- Select the **Setup** application
- Select **Calibrate touch probe**
- Select **Radius** measurement method
- Select the **Calibration sphere** calibration standard
- Enter the diameter of the sphere
- Enter the start angle
- Enter the number of touch points
- Press the **NC Start** key
- > The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- Check results
- Select **Apply calibration data**
- > The control stores the calibrated radius of the 3D touch probe in the tool table.
- > The control shows the **3D calibration** measurement method.
- Select the **3D calibration** measurement method
- Enter the number of touch points
- Press the **NC Start** key
- > The 3D touch probe probes all required touch points in an automatic probing routine.
- Select **Apply calibration data**
- > The control saves the deviations in a compensation value table under **TNC:\system\3D-ToolComp**.
- Select **Exit probing**
- > The control closes the **Calibrate touch probe** function.

Instructions for calibration

- In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.
- If you press the **OK** button after the calibration process, the control accepts the calibration values for the active touch probe. The updated tool data then becomes immediately effective, and it is not necessary to repeat the tool call.
- HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.
- If you want to calibrate using the outside of an object, you need to pre-position the touch probe above the center of the calibration sphere or calibration pin. Ensure that the probing points can be approached without collisions.
- The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset in the touch probe table. The control uses the **TP_NO** parameter to link the data from the touch probe table with the data from the tool table.

  Further information: "Touch probe table tchprobe.tp", Page 1839

28.3 Suppressing touch probe monitoring

Application

If you move a workpiece touch probe too close to the workpiece, you can accidentally deflect the workpiece touch probe. You cannot retract a deflected workpiece touch probe in the monitored state. You can retract a deflected workpiece touch probe by suppressing touch probe monitoring.

Description of function

If the control does not receive a stable signal from the probe, the button displays **Suppress touch probe monitoring**.

As long as touch-probe monitoring is switched off, the control displays the error message **The touch probe monitor is deactivated for 30 seconds**. This error message remains active only for 30 seconds.

28.3.1 Deactivating touch probe monitoring

To deactivate touch probe monitoring:

- ▶ Select the **Manual** operating mode
- ▶ Select **Suppress touch probe monitoring**
- > The control disables touch-probe monitoring for 30 seconds.
- ▶ If required, move the touch probe so that the control receives a stable signal from it.

Notes

NOTICE

Danger of collision!

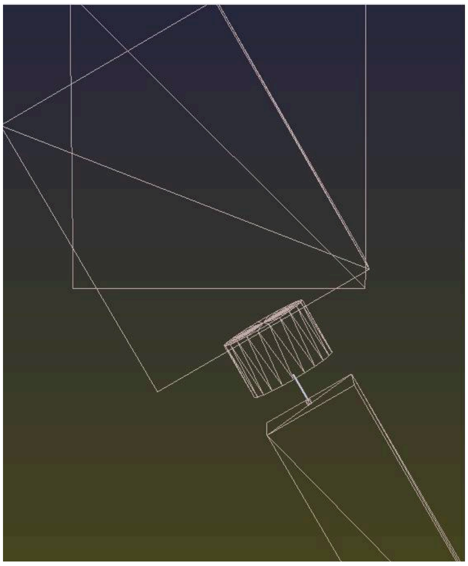
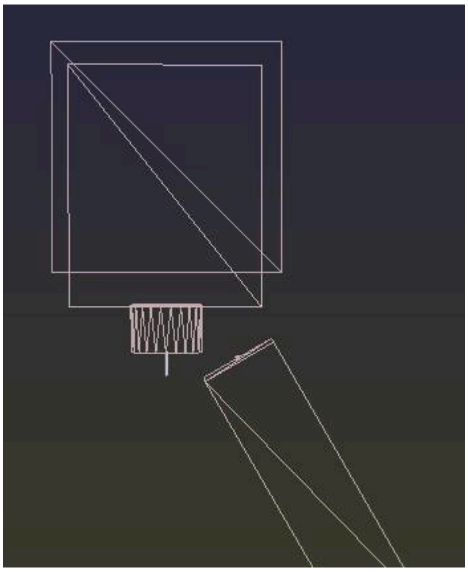
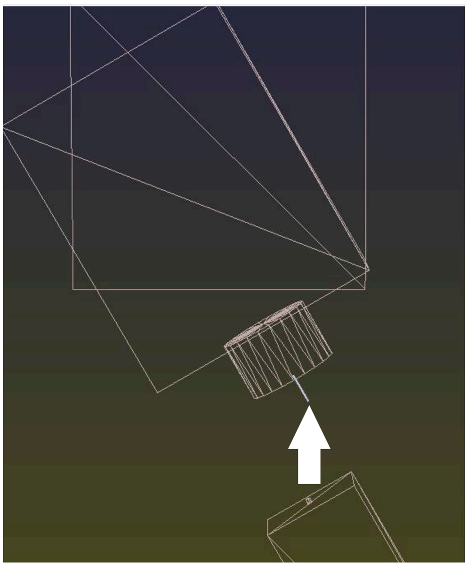
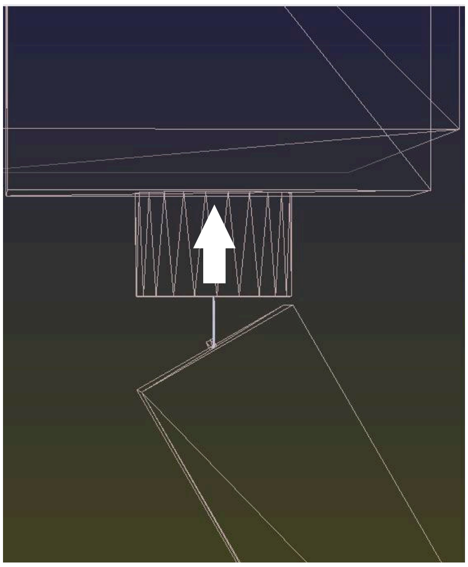
While touch-probe monitoring is deactivated, the control will not perform collision checking. Thus, you must ensure that the touch probe can be positioned safely. There is a risk of collision if you choose the wrong direction of traverse!

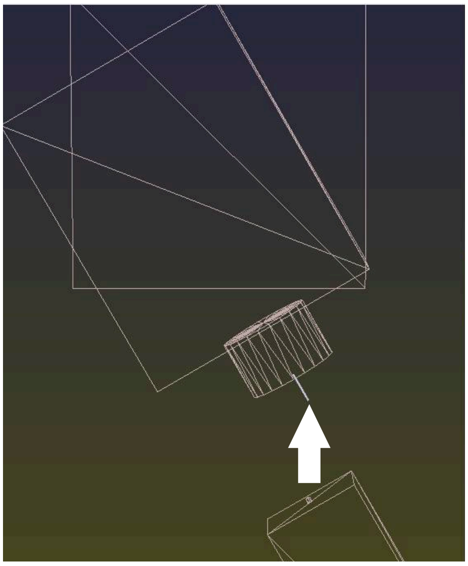
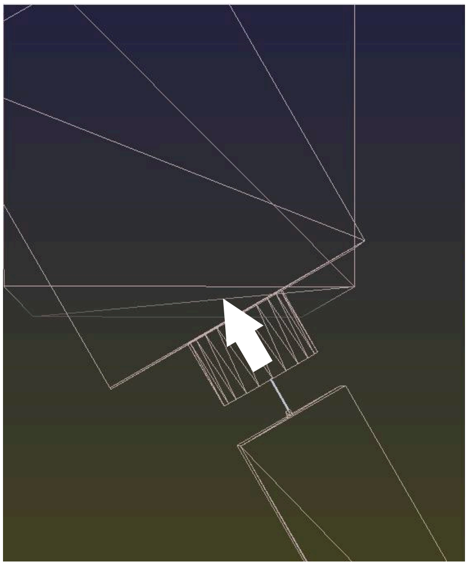
- Carefully move the axes in the **Manual** operating mode

If the touch probe sends a stable signal within the 30 seconds, then touch-probe monitoring reactivates itself automatically and the error message is cleared.

28.4 Comparison of offset and 3D basic rotation

The following example shows how the two functions differ.

Offset	3D basic rotation
Initial state  Position display: ■ Actual position ■ **B** = 0 ■ **C** = 0 Preset table: ■ **SPB** = 0 ■ **B_OFFS** = -30 ■ **C_OFFS** = +0	Initial state  Position display: ■ Actual position ■ **B** = 0 ■ **C** = 0 Preset table: ■ **SPB** = -30 ■ **B_OFFS** = +0 ■ **C_OFFS** = +0
Movement in +Z without tilting 	Movement in +Z without tilting 

Offset	3D basic rotation
Movement in +Z with tilting **PLANE SPATIAL** with **SPA+0 SPB+0 SPC +0**	Movement in +Z with tilting **PLANE SPATIAL** with **SPA+0 SPB+0 SPC +0**
	
> The orientation **is not correct**!	> The orientation is correct! > The next machining step will be **correct**.

HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

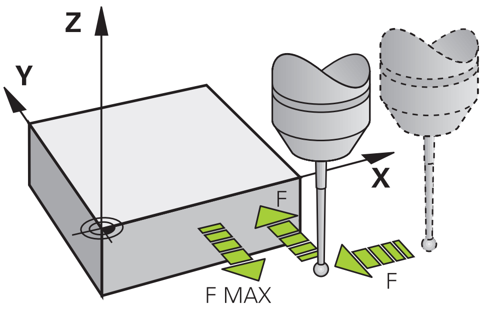
29

Programmable Touch Probe Cycles

29.1 Working with Touch Probe Cycles

29.1.1 General information about touch probe cycles

Method of function



The touch probe functions allow you to set presets on the workpiece, measure the workpiece, and determine and compensate for workpiece misalignment.

Whenever the control runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder will determine the probing feed rate in a machine parameter.

Page 1456

When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the control: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a defined distance, the control displays an error message (distance: **DIST** from touch probe table).

Related topics

- Manual touch probe cycles

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1425
- Preset table

  Further information: "Preset table", Page 1851
- Datum table

  Further information: "Datum table", Page 1858
- Reference systems

  Further information: "Reference systems", Page 934
- Preassigned variables

  Further information: "Preassigned Q parameters", Page 1273

Requirements

- Calibrated workpiece touch probe

  Further information: "Calibrating the workpiece touch probe", Page 1439

  If you are using a HEIDENHAIN touch probe, the software option 17, Touch Probe Functions, is automatically enabled.

Notes

⚙	The control must be specifically prepared by the machine manufacturer for the use of a touch probe. While touch probe functions are being executed, the control temporarily disables the **Global Program Settings**.

i	HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.

Touch probe cycles in the Manual Operation and Electronic Handwheel modes

In the **Setup** application under **Manual** mode, the control provides touch probe cycles that allow you to:

- Set presets
- Probe the angle
- Probe position
- Calibrate the touch probe
- Measure the tool

Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1425

Touch probe cycles for automatic operation

Besides the manual touch probe cycles, several cycles are available for a wide variety of applications in automatic operation:

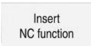
- Automatic measurement of workpiece misalignment
- Automatic determination of the preset
- Automatic workpiece inspection
- Special functions
- Touch probe calibration
- Automatic kinematics measurement
- Automatic tool measurement

Defining touch probe cycles

Like the most recent machining cycles, touch probe cycles with numbers greater than **400** use Q parameters as transfer parameters. Parameters with the same function that the control requires in several cycles always have the same number: For example, **Q260** is always assigned the clearance height, **Q261** the measuring height, etc.

There are various ways to define the touch probe cycles. Touch probe cycles are programmed in the **Programming** mode of operation.

Inserting via NC function:



- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select the desired cycle
- > The control initiates a dialog and prompts you for all required input values.

Inserting via the TOUCH PROBE key:

- Select the **TOUCH PROBE** soft key
- > The control opens the **Insert NC function** window.
- Select the desired cycle
- > The control initiates a dialog and prompts you for all required input values.

Navigation in the cycle

Key	Function
	Navigation within the cycle: Jump to next parameter
	Navigation within the cycle: Jump to previous parameter
	Jump to the same parameter in the next cycle
	Jump to the same parameter in the previous cycle

The control provides selection possibilities for different cycle parameters via the action bar or the form.

Available cycle groups

Machining cycles

Cycle group	Further information
Drilling/Thread	
■ Drilling, reaming	Page 433
■ Boring	Page 452
■ Counterboring, centering	
■ Tapping or thread milling	
Pockets/studs/slots	
■ Pocket milling	Page 452
■ Stud milling	
■ Slot milling	
■ Face milling	
Coordinate transformations	
■ Mirroring	Page 958
■ Rotating	
■ Magnifying / Reducing	
SL cycles	
■ SL (Subcontour List) cycles for the machining of contours that possibly consist of several subcontours	Page 452
■ Cylinder surface machining	Page 1178
■ OCM (Optimized Contour Milling) cycles for combining subcontours to form complex contours	Page 392
Point patterns	
■ Bolt hole circle	Page 377
■ Linear hole pattern	
■ Data Matrix code	
Turning cycles	
■ Area clearance cycles, longitudinal and transverse	Page 677
■ Recess turning cycles, radial and axial	
■ Recessing cycles, radial and axial	
■ Thread cutting cycles	
■ Simultaneous turning cycles	
■ Special cycles	

Cycle group	Further information
Special cycles	
■ Dwell time	Page 1126
■ Program call	Page 452
■ Tolerance	Page 898
■ Oriented spindle stop	Page 1148
■ Engraving	
■ Gear cycles	
■ Interpolation turning	
Grinding cycles	
■ Reciprocating stroke	Page 836
■ Dressing	
■ Correction cycles	

Measuring cycles

Cycle group	Further information
Rotation	
■ Probing of plane, edge, two circles, beveled edge ■ Basic rotation ■ Two holes or studs ■ Via rotary axis ■ Via C-axis	Page 1460
Preset/Position	
■ Rectangle, inside or outside ■ Circle, inside or outside ■ Corner, inside or outside ■ Center of bolt circle, slot or ridge ■ Touch probe axis or single axis ■ Four holes	Page 1524
Measuring	
■ Angle ■ Circle, inside or outside ■ Rectangle, inside or outside ■ Slot or ridge ■ Bolt hole circle ■ Plane or coordinate	Page 1605
Special cycles	
■ Measuring or measuring in 3D ■ Probing in 3D ■ Fast probing	Page 1662
Calibrating the touch probe	
■ Calibrating the length ■ Calibration in a ring ■ Calibration on a stud ■ Calibration on a sphere	Page 1679
Measuring kinematics	
■ Saving the kinematics ■ Measure kinematics ■ Preset compensation ■ Kinematics grid	Page 1695
Measuring the tool (TT)	
■ Calibrating the TT ■ Tool length, radius or measuring completely ■ Calibrating the IR-TT ■ Lathe tool measurement	Page 1734

29.1.2 Before you start working with touch probe cycles!

General information

In the touch probe table you define the set-up clearance, i.e., how far away from the defined touch point (or the one calculated by the cycle), the control will pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to the one from the touch probe table.

The following can be defined in the touch probe table:

- Type of tool
- Touch probe center offset
- Spindle angle during calibration
- Probing feed rate
- Rapid traverse in probing cycle
- Maximum measuring range
- Safety clearance
- Feed rate for pre-positioning
- Touch probe orientation
- Serial number
- Reaction in case of collision

Further information: "Touch probe table tchprobe.tp", Page 1839

Executing touch probe cycles

All touch probe cycles are DEF-active. The control runs the cycle automatically as soon as it reads the cycle definition in the program run.

Positioning logic

Touch probe cycles numbered **400** to **499** or **1400** to **1499** pre-position the touch probe according to the following positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (as defined in the cycle), the control first retracts the touch probe in the touch probe axis to clearance height and then positions it in the working plane to the first touch point.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the control first positions the touch probe in the working plane to the first touch point, and then in the touch-probe axis directly to the set-up clearance.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- ▶ Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- Reset any coordinate transformations before the cycle call.

- Please note that the units of measure in the measuring log and the return parameters depend on the main program.
- The touch probe cycles **40x** to **43x** will reset an active basic rotation at the beginning of the cycle.
- The control interprets a basic transformation as a basic rotation, and an offset as a table rotation.
- You can compensate for the workpiece misalignment by rotating the table only if the machine is designed with a rotary table axis that is oriented perpendicularly with respect to the workpiece coordinate system **W-CS**.

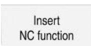
  Further information: "Comparison of offset and 3D basic rotation", Page 1446

Note about machine parameters

- Depending on the setting of the optional machine parameter **chkTiltingAxes** (no. 204600), the control will check during probing whether the position of the rotary axes matches the tilting angles (3D-ROT). If that is not the case, the control displays an error message.

29.1.3 Program defaults for cycles

Entering GLOBAL DEF definitions



- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select **GLOBAL DEF**
- Select the desired **GLOBAL DEF** function, e.g. **100 GENERAL**
- Enter the required definitions

Using GLOBAL DEF information

If you entered the corresponding **GLOBAL DEF** functions at program start, you can reference these globally valid values for the definition of any cycle.

Proceed as follows:

Insert NC function

- Select **Insert NC function**
- > The control opens the **Insert NC function** window.
- Select and define **GLOBAL DEF**
- Select **Insert NC function** again
- Select the desired cycle, e.g. **200 DRILLING**
- > If the cycle includes global cycle parameters, the control superimposes the selection possibility **PREDEF** in the action bar or in the form as a selection menu.

PREDEF

- Select **PREDEF**
- > The control then enters the word **PREDEF** in the cycle definition. This creates a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program.

NOTICE

Danger of collision!

If you later edit the program settings with **GLOBAL DEF**, these changes will affect the entire NC program. This may change the machining sequence significantly.

- Make sure to use **GLOBAL DEF** carefully. Simulate your program before executing it
- If you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

Global data valid everywhere

The parameters are valid for all **2xx** machining cycles as well as for Cycles **880, 1017**, **1018**, **1021**, **1022**, **1025** and touch probe cycles **451**, **452**, **453**

Help graphic	Parameter
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: **0...99999.9999**
	Q204 2nd set-up clearance? Distance in the tool axis between touch probe and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Feed rate at which the control moves the tool within a cycle. Input: **0...99999.999** or **FMAX**, **FAUTO**
	Q208 Feed rate for retraction? Feed rate at which the control retracts the tool. Input: **0...99999.999** or **FMAX**, **FAUTO**

Example

```
11 GLOBAL DEF 100 GENERAL ~
    Q200=+2                  ;SET-UP CLEARANCE ~
    Q204=+50                 ;2ND SET-UP CLEARANCE ~
    Q253=+750                ;F PRE-POSITIONING ~
    Q208=+999                ;RETRACTION FEED RATE
```

Global data for probing functions

The parameters are valid for all touch probe cycles **4xx** and **14xx** as well as for Cycles **271**, **286**, **287**, **880**, **1021**, **1022**, **1025**, **1271**, **1272**, **1273**, **1278**

Help graphic	Parameter
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**

Example

```
11 GLOBAL DEF 120 PROBING ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q301=+1                 ;MOVE TO CLEARANCE
```

29.2 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

29.2.1 Overview

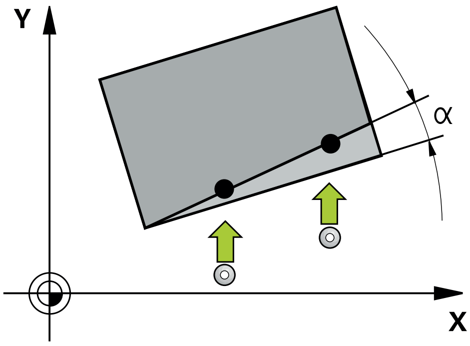
> The control must be specifically prepared by the machine manufacturer for the use of a touch probe.
>
> HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.

Cycle	Call	Further information
1420 PROBING IN PLANE ■ Automatic measurement using three points ■ Compensation via basic rotation or rotary table rotation	**DEF**-active	Page 1472

Cycle		Call	Further information
1410	**PROBING ON EDGE** ■ Automatic measurement using two points ■ Compensation via basic rotation or rotary table rotation	**DEF**-active	Page 1478
1411	**PROBING TWO CIRCLES** ■ Automatic measurement using two holes or studs ■ Compensation via basic rotation or rotary table rotation	**DEF**-active	Page 1485
1412	**INCLINED EDGE PROBING** ■ Automatic measurement using two points on an inclined edge ■ Compensation via basic rotation or rotary table rotation	**DEF**-active	Page 1493
400	**BASIC ROTATION** ■ Automatic measurement using two points ■ Compensation via basic rotation	**DEF**-active	Page 1500
401	**ROT OF 2 HOLES** ■ Automatic measurement using two holes ■ Compensation via basic rotation	**DEF**-active	Page 1502
402	**ROT OF 2 STUDS** ■ Automatic measurement using two studs ■ Compensation via basic rotation	**DEF**-active	Page 1506
403	**ROT IN ROTARY AXIS** ■ Automatic measurement using two points ■ Compensation via rotary table rotation	**DEF**-active	Page 1511
405	**ROT IN C AXIS** ■ Automatic alignment of an angular offset between a hole center and the positive Y axis ■ Compensation via rotary table rotation	**DEF**-active	Page 1517
404	**SET BASIC ROTATION** ■ Setting any basic rotation	**DEF**-active	Page 1522

29.2.2 Fundamentals of touch probe cycles 14xx

Functionalities common to touch probe cycles 14xx for measuring rotations



The cycles are capable of determining rotation and include the following:

- Consideration of active machine kinematics
- Semi-automatic probing
- Monitoring of tolerances
- Consideration of 3-D calibration
- Simultaneous measurement of rotation and position

Programming notes:

- The probing positions are referenced to the programmed nominal coordinates in the I-CS.
- See the drawing for these nominal positions.
- Before a cycle definition you must program a tool call to define the touch-probe axis.

Explanation of terms

Designation	Short description
Nominal position	Position in the drawing (e.g., position of a hole)
Nominal dimension	Dimension in the drawing (e.g., hole diameter)
Actual position	Measured position (e.g., position of a hole)
Actual dimension	Measured dimension (e.g., hole diameter)
I-CS	I-CS: **Input Coordinate System**
W-CS	W-CS: **Workpiece Coordinate System**
Object	Object to be probed: circle, stud, plane, edge
Surface-normal vectors	

Evaluation – preset:

- If you want to probe objects in a consistent machining plane or probe objects while TCPM is active, you can program any required shifts as basic transformations in the preset table.
- Rotations can be written to the basic transformations of the preset table as basic rotations or as axial offsets from the first rotary table axis, seen from the workpiece.

> Operating notes:
> - When probing, existing 3-D calibration data are taken into account. If these calibration data do not exist, deviations might be the result.
> - If you want to use not only the measured rotation, but also a measured position, make sure to probe the surface perpendicularly, if possible. The larger the angular error and the bigger the ball-tip radius, the larger the positioning error. If the angular errors in the initial angular position are too large, corresponding position errors might be the result.

Logging:

The measured results are recorded in the **TCHPRAUTO.html** file and stored in the Q parameters programmed for this cycle.

The measured deviations are the differences between the measured actual values and the mean tolerance value. If no tolerance has been specified, they refer to the nominal dimension.

The unit of measurement of the main program can be seen in the header of the log.

Semi-automatic mode

If the probing positions relative to the current datum are unknown, you can execute the cycle in semi-automatic mode. In this mode, you can determine the starting position by manually pre-positioning before performing the probing operation.

For this purpose, precede the value for the required nominal position with **"?"**. You can do this by selecting **Name** in the action bar. Depending on the object, you need to define the nominal positions that determine the probing direction, see "Examples."

> Depending on the object, you need to define the nominal positions that determine the probing direction,
>
> Examples:
>
> - Page 1465
> - Page 1466
> - Page 1467
>

Cycle sequence

Proceed as follows:

- ▶ Run the cycle
- > The control interrupts the NC program.
- > A window opens.
- ▶ Use the axis-direction keys to position the touch probe to the desired touch point

  or
- ▶ Position the touch probe to the desired point using the electronic handwheel
- ▶ Change the probing direction in the window, if necessary
- ▶ Select the **NC start** key
- > The control closes the window and performs the first probing operation.
- > If **CLEAR. HEIGHT MODE Q1125** = **1** or **2**, the control opens a message in the **FN 16** tab, **Status** workspace, explaining that the Traverse to clearance height mode cannot be used here.
- ▶ Move the tool to a safe position
- ▶ Select the **NC start** key
- > Cycle or program execution is resumed. You may then need to repeat the entire process for further touch points.

NOTICE

Danger of collision!

The control will ignore the programmed values 1 and 2 for Traverse to clearance height when running in semi-automatic mode. Depending on the position of the touch probe, there is danger of collision.

- ▶ In semi-automatic mode, traverse to clearance height after each probing operation.

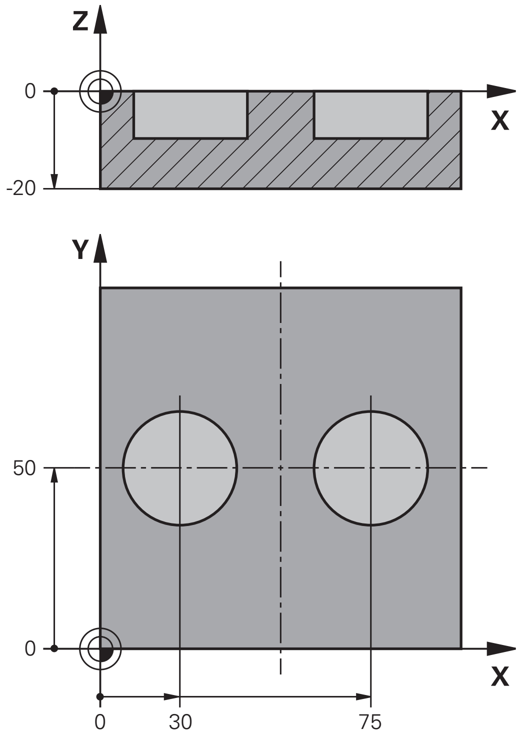
Programming and operating notes:

- See the drawing for these nominal positions.
- Semi-automatic mode is only executed in the machine operating modes, not in the simulation.
- If you did not define a nominal position for a touch point in any direction, the control generates an error message.
- If you did not define a nominal position for a single direction, the control will capture the actual position after probing the object. This means that the measured actual position will subsequently be applied as the nominal position. Consequentially, there is no deviation for this position and thus no position compensation.

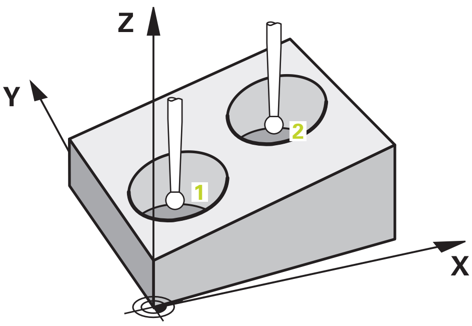
Examples

Important: Specify the **nominal positions** from the drawing!

In the following three examples, the nominal positions from this drawing will be used.



Alignment using two holes



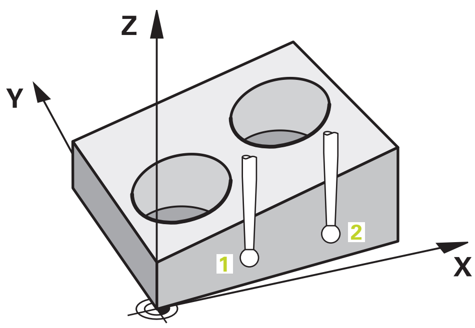
In this example, you will align two holes. Probing is done in the X axis (main axis) and in the Y axis (secondary axis). This means that it is mandatory to define the nominal position from the drawing for these axes! A nominal position for the Z axis (tool axis) is not necessary as you will not measure in this direction.

- **QS1100** = nominal position 1 in main axis is defined, but workpiece position is unknown
- **QS1101** = nominal position 1 in secondary axis is defined, but workpiece position is unknown
- **QS1102** = nominal position 1 in tool axis is unknown
- **QS1103** = nominal position 2 in main axis is defined, but workpiece position is unknown

- **QS1104** = nominal position 2 in secondary axis is defined, but workpiece position is unknown
- **QS1105** = nominal position 2 in tool axis is unknown

11 TCH PROBE 1411 PROBING TWO CIRCLES ~	
QS1100= "?30"	;1ST POINT REF AXIS ~
QS1101= "?50"	;1ST POINT MINOR AXIS ~
QS1102= "?"	;1ST POINT TOOL AXIS ~
Q1116=+10	;DIAMETER 1 ~
QS1103= "?75"	;2ND POINT REF AXIS ~
QS1104= "?50"	;2ND POINT MINOR AXIS ~
QS1105= "?"	;2ND POINT TOOL AXIS ~
Q1117=+10	;DIAMETER 2 ~
Q1115=+0	;GEOMETRY TYPE ~
Q423=+4	;NO. OF PROBE POINTS ~
Q325=+0	;STARTING ANGLE ~
Q1119=+360	;ANGULAR LENGTH ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

Alignment through an edge



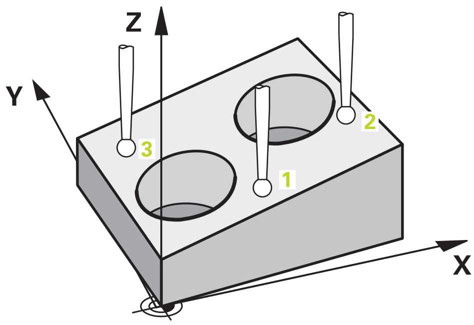
In this example, you will align an edge. Probing is done in the Y axis (secondary axis). This means that it is mandatory to define the nominal position from the drawing for these axes! Nominal positions for the X axis (main axis) and for the Z axis (tool axis) are not required because you will not measure in these directions.

- **QS1100** = nominal position 1 in main axis is unknown
- **QS1101** = nominal position 1 in secondary axis is defined, but workpiece position is unknown
- **QS1102** = nominal position 1 in tool axis is unknown
- **QS1103** = nominal position 2 in main axis is unknown

- **QS1104** = nominal position 2 in secondary axis is defined, but workpiece position is unknown
- **QS1105** = nominal position 2 in tool axis is unknown

11 TCH PROBE 1410 PROBING ON EDGE ~	
QS1100= "?"	;1ST POINT REF AXIS ~
QS1101= "?0"	;1ST POINT MINOR AXIS ~
QS1102= "?"	;1ST POINT TOOL AXIS ~
QS1103= "?"	;2ND POINT REF AXIS ~
QS1104= "?0"	;2ND POINT MINOR AXIS ~
QS1105= "?"	;2ND POINT TOOL AXIS ~
Q372=+2	;PROBING DIRECTION ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

Alignment via the plane



In this example, you will align a plane. In this case, it is mandatory to define all three nominal positions from the drawing. For angle calculations, it is important that all three axes are taken into account when probing.

- **QS1100** = nominal position 1 in main axis is defined, but workpiece position is unknown
- **QS1101** = nominal position 1 in secondary axis is defined, but workpiece position is unknown
- **QS1102** = nominal position 1 in tool axis is defined, but workpiece position is unknown
- **QS1103** = nominal position 2 in main axis is defined, but workpiece position is unknown
- **QS1104** = nominal position 2 in secondary axis is defined, but workpiece position is unknown
- **QS1105** = nominal position 2 in tool axis is defined, but workpiece position is unknown
- **QS1106** = nominal position 3 in main axis is defined, but workpiece position is unknown

- **QS1107** = nominal position 3 in secondary axis is defined, but workpiece position is unknown
- **QS1108** = nominal position 3 in tool axis is defined, but workpiece position is unknown

11 TCH PROBE 1420 PROBING IN PLANE ~		; Define the cycle
QS1100= "?50"	**;1ST POINT REF AXIS ~**	; Nominal position 1 in main axis is defined, but workpiece position is unknown
QS1101= "?10"	**;1ST POINT MINOR AXIS ~**	; Nominal position 1 in secondary axis is defined, but workpiece position is unknown
QS1102= "?0"	**;1ST POINT TOOL AXIS ~**	; Nominal position 1 in tool axis is defined, but workpiece position is unknown
QS1103= "?80"	**;2ND POINT REF AXIS ~**	; Nominal position 2 in main axis is defined, but workpiece position is unknown
QS1104= "?50"	**;2ND POINT MINOR AXIS ~**	; Nominal position 2 in secondary axis is defined, but workpiece position is unknown
QS1105= "?0"	**;2ND POINT TOOL AXIS ~**	; Nominal position 2 in tool axis is defined, but workpiece position is unknown
QS1106= "?20"	**;3RD POINT REF AXIS ~**	; Nominal position 3 in main axis is defined, but workpiece position is unknown
QS1107= "?80"	**;3RD POINT MINOR AXIS ~**	; Nominal position 3 in secondary axis is defined, but workpiece position is unknown
QS1108= "?0"	**;3RD POINT TOOL AXIS ~**	; Nominal position 3 in tool axis is defined, but workpiece position is unknown
Q372=-3	**;PROBING DIRECTION ~**	; Probing direction Z–
Q320=+2	**;SET-UP CLEARANCE ~**	
Q260=+100	**;CLEARANCE HEIGHT ~**	
Q1125=+2	**;CLEAR. HEIGHT MODE ~**	
Q309=+0	**;ERROR REACTION ~**	
Q1126=+0	**;ALIGN ROTARY AXIS ~**	
Q1120=+0	**;TRANSER POSITION ~**	
Q1121=+0	**;CONFIRM ROTATION**	

Evaluation of tolerances

Cycles 14xx also allow you to check tolerance bands. This includes the checking of the position and size of an object.

The following input values with tolerances are possible:

Tolerance	Example
Dimensions	10+0.01-0.015
DIN EN ISO 286-2	10H7
ISO 2768-1	10m

Pay attention to capitalization when entering tolerances.

If you enter a value with a tolerance in your program, the control will monitor the tolerance band. The control writes the following statuses to the return parameter **Q183**: Pass, rework, or scrap. If a compensation of the preset is programmed, the control corrects the active preset after probing

The following cycle parameters allow input values with tolerances:

- **Q1100 1ST POINT REF AXIS**
- **Q1101 1ST POINT MINOR AXIS**
- **Q1102 1ST POINT TOOL AXIS**
- **Q1103 2ND POINT REF AXIS**
- **Q1104 2ND POINT MINOR AXIS**
- **Q1105 2ND POINT TOOL AXIS**
- **Q1106 3RD POINT REF AXIS**
- **Q1107 3RD POINT MINOR AXIS**
- **Q1108 3RD POINT TOOL AXIS**
- **Q1116 DIAMETER 1**
- **Q1117 DIAMETER 2**

Program this as follows:

- Start the cycle definition
- Enable the Name selection option in the action bar
- Program nominal position/dimension incl. tolerance
- > In the cycle, **QS1116="+8-2-1"** is defined, for example.

If you program an incorrect tolerance, the control will interrupt machining with an error message.

Cycle sequence

If the actual position is outside the tolerance, the control behaves as follows:

- **Q309 = 0**: The control does not interrupt program run.
- **Q309 = 1**: In the case of scrap or rework, the control interrupts program run with a message.
- **Q309 = 2**: In the case of scrap, the control interrupts program run with a message.

If Q309 = 1 or 2, proceed as follows:

> A window appears. The control displays all of the nominal and actual dimensions of the object.

- Press the **CANCEL** button to interrupt NC program run

or

- Press **NC start** to resume NC program run

> Please note that the deviations returned by the touch probe cycles are with respect to the mean tolerance in **Q98x** and **Q99x**. The values are thus the same as the correction values executed by the cycle if the input parameters **Q1120** and **Q1121** are programmed. If no automatic evaluation is active, the control will save the values with respect to the mean tolerance in the programmed Q parameters. You can further process these values.

Example

- QS1116 = diameter 1, tolerance specified
- QS1117 = diameter 2, tolerance specified

11 TCH PROBE 1411PROBING TWO CIRCLES ~	
Q1100=+30	**;1ST POINT REF AXIS ~**
Q1101=+50	**;1ST POINT MINOR AXIS ~**
Q1102=-5	**;1ST POINT TOOL AXIS ~**
QS1116="+8-2-1"	**;DIAMETER 1 ~**
Q1103=+75	**;2ND POINT REF AXIS ~**
Q1104=+50	**;2ND POINT MINOR AXIS ~**
QS1105=-5	**;2ND POINT TOOL AXIS ~**
QS1117="+8-2-1"	**;DIAMETER 2 ~**
Q1115=+0	**;GEOMETRY TYPE ~**
Q423=+4	**;NO. OF PROBE POINTS ~**
Q325=+0	**;STARTING ANGLE ~**
Q1119=+360	**;ANGULAR LENGTH ~**
Q320=+2	**;SET-UP CLEARANCE ~**
Q260=+100	**;CLEARANCE HEIGHT ~**
Q1125=+2	**;CLEAR. HEIGHT MODE ~**
Q309=2	**;ERROR REACTION ~**
Q1126=+0	**;ALIGN ROTARY AXIS ~**
Q1120=+0	**;TRANSER POSITION ~**
Q1121=+0	**;CONFIRM ROTATION**

Transferring the actual position

You can determine the actual position in advance and define it as the actual position for the touch probe cycle. Then, both the nominal position and the actual position will be transferred to the object. Based on the difference, the cycle calculates the required compensation values and applies tolerance monitoring.

Program this as follows:

- Define the cycle
- Enable the Name selection option in the action bar
- Program nominal position incl. tolerance, if necessary
- Program **"@"**
- Program actual position
- > In the cycle, **QS1100="10+0.02@10.0123"** is defined, for example.

Programming and operating notes:

- If you program @, no probing will be carried out. The control only accounts for the actual and nominal positions.
- You must define the actual position for all three axes: main axis, secondary axis, and tool axis. If you define only one axis with its actual position, an error message will be generated.
- Actual positions can also be defined with Q **Q1900-Q1999**

Example

This feature allows you to do the following:

- Determine a circular pattern based on multiple different objects
- Align a gear wheel via its center and the position of a tooth

The nominal positions are defined here with tolerance monitoring and actual position.

```
5 TCH PROBE 1410 PROBING ON EDGE ~
    QS1100="10+0.02@10.0123"    ;1ST POINT REF AXIS ~
    QS1101="50@50.0321"         ;1ST POINT MINOR AXIS ~
    QS1102="-10-0.2+0.2@Q1900"  ;1ST POINT TOOL AXIS ~
    QS1103="30+0.02@30.0134"    ;2ND POINT REF AXIS ~
    QS1104="50@50.534"          ;2ND POINT MINOR AXIS ~
    QS1105="-10-0.02@Q1901"     ;2ND POINT TOOL AXIS ~
    Q372=+2                     ;PROBING DIRECTION ~
    Q320=+0                     ;SET-UP CLEARANCE ~
    Q260=+100                   ;CLEARANCE HEIGHT ~
    Q1125=+2                    ;CLEAR. HEIGHT MODE ~
    Q309=+0                     ;ERROR REACTION ~
    Q1126=+0                    ;ALIGN ROTARY AXIS ~
    Q1120=+0                    ;TRANSER POSITION ~
    Q1121=+0                    ;CONFIRM ROTATION
```

29.2.3 Cycle 1420 PROBING IN PLANE

Application

Touch probe cycle **1420** finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1676

Furthermore, you can perform the following operations with Cycle **1420**:

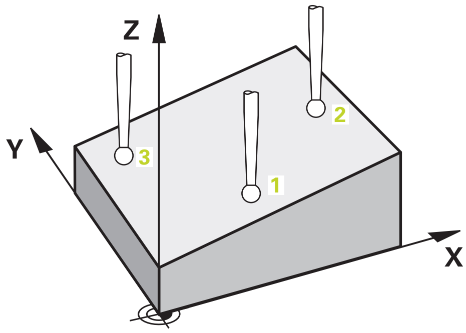
- If the probing position relative to the current datum is unknown, you can execute the cycle in semi-automatic mode.

  Page 1463
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

  Page 1469
- If you determine the actual position in advance, you can transfer it to the cycle as the actual position.

  Page 1471

Cycle sequence



1. Following the positioning logic, the control positions the touch probe to the programmed touch point **1** at rapid traverse **FMAX_PROBE**.

   Further information: "Positioning logic", Page 1456
2. The control then moves the touch probe to set-up clearance at rapid traverse **FMAX_PROBE**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account when probing in any probing direction.
3. Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate **F** from the touch probe table.
4. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
5. If you programmed retraction to clearance height **Q1125**, the touch probe returns to clearance height.
6. It then moves in the working plane to touch point **2** to measure the actual value of the second touch point in the plane.
7. The touch probe returns to the clearance height (depending on **Q1125**), then moves in the working plane to touch point **3** and measures the actual position of the third point of the plane.
8. Finally, the control retracts the touch probe to the clearance height (depending on **Q1125**) and saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to **Q955**	Measured position 2 in the main axis, secondary axis, and tool axis
Q956 to **Q958**	Measured position 3 in the main axis, secondary axis, and tool axis
Q961 to **Q963**	Measured spatial angle SPA, SPB, and SPC in the W-CS
Q980 to **Q982**	Measured deviations of touch point 1
Q983 to **Q985**	Measured deviations of touch point 2
Q986 to **Q988**	Deviation 3 of the positions
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Mean value of all deviations from the ideal line of the first touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Mean value of all deviations from the ideal line of the second touch point
Q972	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Mean value of all deviations from the ideal line of the third touch point

Notes

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

- Always move to clearance height between objects or touch points

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three touch points are not positioned on a straight line.
- The nominal spatial angle results from the defined nominal positions. The control saves the calculated spatial angle in parameters **Q961** to **Q963**. The control transfers the difference between the measured spatial angle and the nominal spatial angle to the basic rotation in 3-D of the preset table.

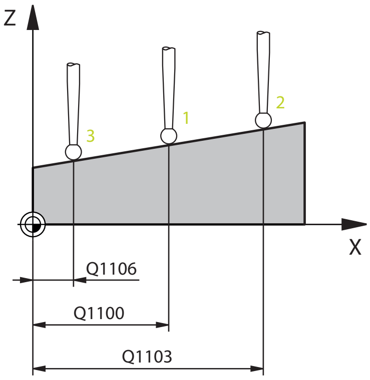
- HEIDENHAIN recommends avoiding the use of axis angles in this cycle!

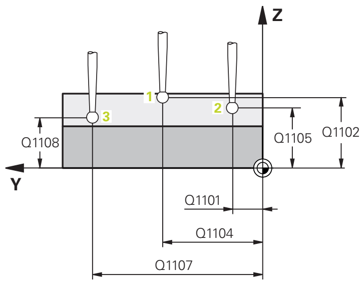
Aligning the rotary table axes:

- Alignment with rotary table axes is possible only if two rotary table axes have been defined in the kinematics..
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be used (**Q1121** not equal to 0). Otherwise, an error message will be displayed.

Cycle parameters

Help graphic





Parameter

Q1100 1st noml. position of ref. axis?

Absolute nominal position of the first touch point in the main axis of the working plane

Input: **-99999.9999...+99999.9999** or optionally **?, -, +**, **@**

- **?**: Semiautomatic mode, Page 1463
- **-, +**: Evaluation of the tolerance, Page 1469
- **@**: Transferring the actual position, Page 1471

Q1101 1st noml. position of minor axis?

Absolute nominal position of the first touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1103 2nd noml. position of ref axis?

Absolute nominal position of the second touch point in the main axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1104 2nd noml. position of minor axis?

Absolute nominal position of the second touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1105 2nd nominal pos. of tool axis?

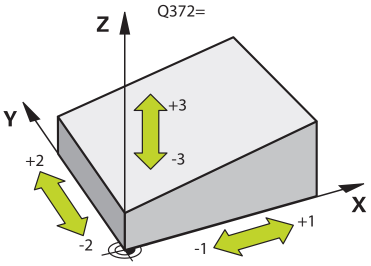
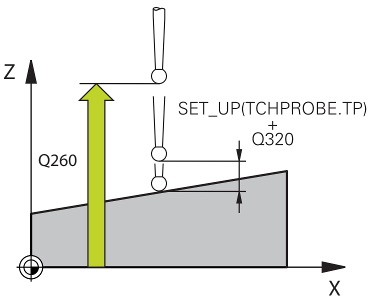
Absolute nominal position of the second touch point in the tool axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1106 3rd noml. position of ref axis?

Absolute nominal position of the third touch point in the main axis of the working plane.

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Help graphic	Parameter
	Q1107 3rd noml. position minor axis? Absolute nominal position of the third touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1108 3rd nominal position tool axis? Absolute nominal position of the third touch point in the tool axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	**Q372 Probe direction (-3 to +3)?** Axis defining the direction of probing. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis. Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**
	**Q320 Set-up clearance?** Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to clearance height. **0**: Move to clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**

Help graphic	Parameter
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary. Input: **0**, **1**, **2**
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: **0**: Retain the current position of the rotary axis. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. **2**: Automatically position the rotary axis without orienting the tool tip (**TURN**). Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point **2**: Correction based on the 2nd touch point **3**: Correction based on the 3rd touch point **4**: Correction based on the averaged touch point position Input: **0**, **1**, **2**, **3**, **4**
	Q1121 Confirm basic rotation? Define whether the control will use the determined misalignment as a basic rotation: **0**: No basic rotation **1**: Set basic rotation: The control will save the basic rotation Input: **0**, **1**

Example

11 TCH PROBE 1420 PROBING IN PLANE ~	
Q1100=+0	;1ST POINT REF AXIS ~
Q1101=+0	;1ST POINT MINOR AXIS ~
Q1102=+0	;1ST POINT TOOL AXIS ~
Q1103=+0	;2ND POINT REF AXIS ~
Q1104=+0	;2ND POINT MINOR AXIS ~
Q1105=+0	;2ND POINT TOOL AXIS ~
Q1106=+0	;3RD POINT REF AXIS ~
Q1107=+0	;3RD POINT MINOR AXIS ~
Q1108=+0	;3RD POINT MINOR AXIS ~
Q372=+1	;PROBING DIRECTION ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

29.2.4 Cycle 1410 PROBING ON EDGE

Application

Touch probe cycle **1410** allows you to determine workpiece misalignment by probing two points on an edge. The cycle determines the rotation based on the difference between the measured angle and the nominal angle.

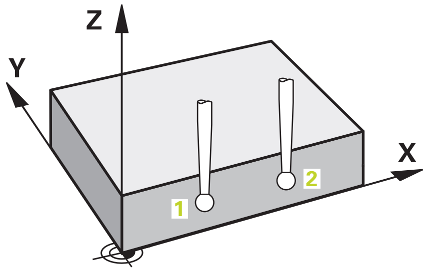
If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1676

Furthermore, you can perform the following operations with Cycle **1410**:

- If the probing position relative to the current datum is unknown, you can execute the cycle in semi-automatic mode.
  Page 1463
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.
  Page 1469
- If you determine the actual position in advance, you can transfer it to the cycle as the actual position.
  Page 1471

Cycle sequence



1. Following the positioning logic, the control positions the touch probe to the programmed touch point **1** at rapid traverse **FMAX_PROBE**.

   Further information: "Positioning logic", Page 1456
2. The control then moves the touch probe to set-up clearance at rapid traverse **FMAX_PROBE**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account when probing in any probing direction.
3. Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate **F** from the touch probe table.
4. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
5. If you programmed retraction to clearance height **Q1125**, the touch probe returns to clearance height.
6. The touch probe then moves to the next touch point **2** and probes again.
7. Finally, the control retracts the touch probe to the clearance height (depending on **Q1125**) and saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to **Q955**	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to **Q982**	Measured deviations of touch point 1
Q983 to **Q985**	Measured deviations of touch point 2
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status ▪ **-1** = Not defined ▪ **0** = Good ▪ **1** = Rework ▪ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Mean value of all deviations from the ideal line of the first touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Mean value of all deviations from the ideal line of the second touch point

Notes

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

- Always move to clearance height between objects or touch points

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Note about rotary axes:

When determining the basic rotation in a tilted working plane, keep the following in mind:

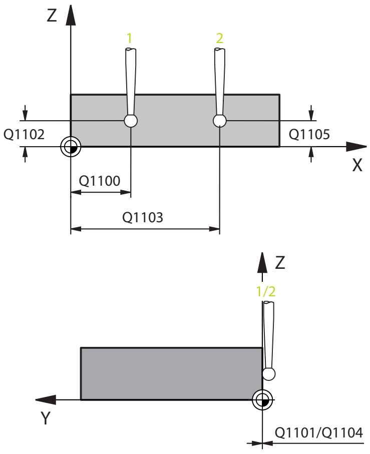
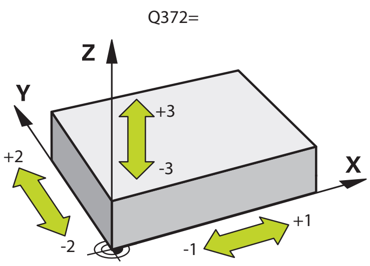
- If the current coordinates of the rotary axes and the defined tilt angles (3-D rotation window) match, the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
- If the current coordinates of the rotary axes and the defined tilt angles (3-D rotation window) do not match, the working plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS**, based on the tool axis.

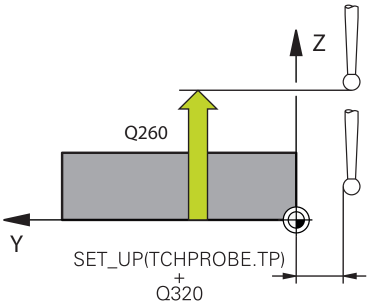
In the optional machine parameter **chkTiltingAxes** (no. 204601), the machine manufacturer defines a check verifying the tilting situation. If no checking is configured, the cycle always assumes that the working plane is consistent. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you need to adopt the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the first touch point in the main axis of the working plane Input: **-99999.9999...+99999.9999** or optionally **?, -, +**, **@** ■ **?**: Semiautomatic mode, Page 1463 ■ **-, +**: Evaluation of the tolerance, Page 1469 ■ **@**: Transferring the actual position, Page 1471
	Q1101 1st noml. position of minor axis? Absolute nominal position of the first touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1103 2nd noml. position of ref axis? Absolute nominal position of the second touch point in the main axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1104 2nd noml. position of minor axis? Absolute nominal position of the second touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1105 2nd nominal pos. of tool axis? Absolute nominal position of the second touch point in the tool axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	**Q372 Probe direction (-3 to +3)?** Axis defining the direction of probing. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis. Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**

Help graphic	Parameter
	**Q320 Set-up clearance?** Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to clearance height. **0**: Move to clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary. Input: **0**, **1**, **2**

Help graphic	Parameter
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: **0**: Retain the current position of the rotary axis. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. **2**: Automatically position the rotary axis without orienting the tool tip (**TURN**). Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point **2**: Correction based on the 2nd touch point **3**: Correction based on the averaged touch point position Input: **0**, **1**, **2**, **3**
	Q1121 CONFIRM ROTATION? Define whether the control will use the determined misalignment as a basic rotation: **0**: No basic rotation **1**: Set basic rotation: The control transfers the misalignment to the preset table as basic transformations. **2**: Rotate the rotary table: The control transfers the misalignment to the preset table as offset. Input: **0**, **1**, **2**

Example

```
11 TCH PROBE 1410 PROBING ON EDGE ~
    Q1100=+0              ;1ST POINT REF AXIS ~
    Q1101=+0              ;1ST POINT MINOR AXIS ~
    Q1102=+0              ;1ST POINT TOOL AXIS ~
    Q1103=+0              ;2ND POINT REF AXIS ~
    Q1104=+0              ;2ND POINT MINOR AXIS ~
    Q1105=+0              ;2ND POINT TOOL AXIS ~
    Q372=+1               ;PROBING DIRECTION ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q260=+100             ;CLEARANCE HEIGHT ~
    Q1125=+2              ;CLEAR. HEIGHT MODE ~
    Q309=+0               ;ERROR REACTION ~
    Q1126=+0              ;ALIGN ROTARY AXIS ~
    Q1120=+0              ;TRANSER POSITION ~
    Q1121=+0              ;CONFIRM ROTATION
```

29.2.5 Cycle 1411 PROBING TWO CIRCLES

Application

Touch probe cycle **1411** captures the center points of two holes or cylindrical studs and calculates a straight line connecting these center points. The cycle determines the rotation in the working plane based on the difference between the measured angle and the nominal angle.

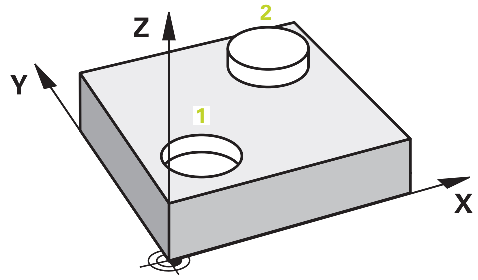
If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1676

Furthermore, you can perform the following operations with Cycle **1411**:

- If the probing position relative to the current datum is unknown, you can execute the cycle in semi-automatic mode.
  Page 1463
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.
  Page 1469
- If you determine the actual position in advance, you can transfer it to the cycle as the actual position.
  Page 1471

Cycle sequence



1. Following the positioning logic, the control positions the touch probe to the programmed center point **1** at the feed rate (depending on **Q1125**).

   Further information: "Positioning logic", Page 1456
2. The control then moves the touch probe to set-up clearance at rapid traverse **FMAX_PROBE**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account when probing in any probing direction.
3. Then the touch probe moves to the entered measuring height at the probing feed rate **F** from the touch probe table and probes (the number of touch points to be probed depends on **Q423** Number of Probes) the first hole or stud center point.
4. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
5. The touch probe returns to the clearance height and then to the position entered as center of the second hole or second stud **2**.
6. Then the control moves the touch probe to the entered measuring height and probes (the number of touch points to be probed depends on **Q423** Number of Probes) the second hole or stud center point.
7. Finally, the control retracts the touch probe to the clearance height (depending on **Q1125**) and saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q950 to **Q952**	Measured circle center point 1 in the main axis, secondary axis, and tool axis
Q953 to **Q955**	Measured circle center point 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q966 to **Q967**	Measured first and second diameters
Q980 to **Q982**	Measured deviations of circle center point 1
Q983 to **Q985**	Measured deviations of circle center point 2
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q996 to **Q997**	Measured deviation of the diameters
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Mean value of all deviations from the ideal line of the first circle center point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING**: Mean value of all deviations from the ideal line of the second circle center point
Q973	If you have programmed Cycle **1493 EXTRUSION PROBING**: Mean value of all deviations of the diameters of circle 1
Q974	If you have programmed Cycle **1493 EXTRUSION PROBING**: Mean value of all deviations of the diameters of circle 2

Operating note:

- If the hole is too small to achieve the programmed set-up clearance, a window opens. In the window, the control displays the nominal dimension of the hole, the calibrated ball-tip radius, and the achievable set-up clearance.

  You have the following options:
  - If there is no danger of collision, you can press NC start to run the cycle with the values from the dialog. The effective set-up clearance will be reduced to the displayed value for this object only.
  - You can cancel the cycle by pressing Cancel.

Notes

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

- Always move to clearance height between objects or touch points

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Note about rotary axes:

When determining the basic rotation in a tilted working plane, keep the following in mind:

- If the current coordinates of the rotary axes and the defined tilt angles (3-D rotation window) match, the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
- If the current coordinates of the rotary axes and the defined tilt angles (3-D rotation window) do not match, the working plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS**, based on the tool axis.

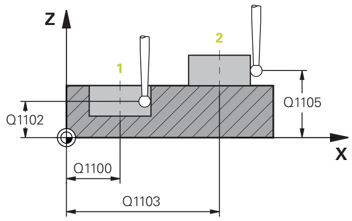
In the optional machine parameter **chkTiltingAxes** (no. 204601), the machine manufacturer defines a check verifying the tilting situation. If no checking is configured, the cycle always assumes that the working plane is consistent. The basic rotation is then calculated in the **I-CS**.

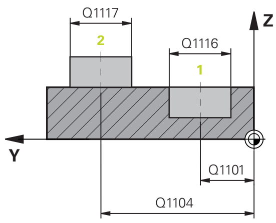
Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you need to adopt the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

Cycle parameters

Help graphic | **Parameter**





Q1100 1st noml. position of ref. axis?

Absolute nominal position of the first touch point in the main axis of the working plane

Input: **-99999.9999...+99999.9999** or optionally **?, -, +**, **@**

- **?**: Semiautomatic mode, Page 1463
- **-, +**: Evaluation of the tolerance, Page 1469
- **@**: Transferring the actual position, Page 1471

Q1101 1st noml. position of minor axis?

Absolute nominal position of the first touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

Absolute nominal position of the first touch point in the tool axis

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1116 Diameter of 1st position?

Diameter of the first hole or the first stud

Input: **0...9999.9999** or optional input:

"...-...+...": Evaluation of the tolerance, Page 1469

Q1103 2nd noml. position of ref axis?

Absolute nominal position of the second touch point in the main axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1104 2nd noml. position of minor axis?

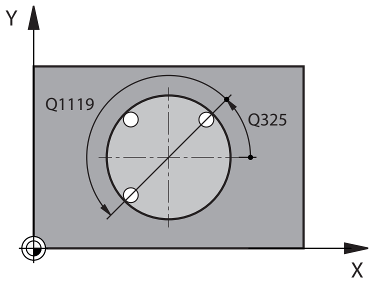
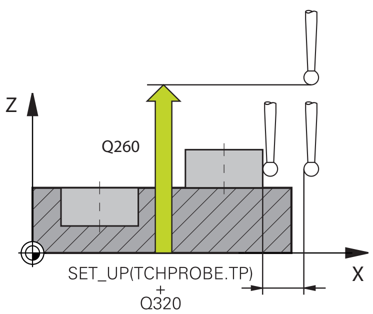
Absolute nominal position of the second touch point in the secondary axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Q1105 2nd nominal pos. of tool axis?

Absolute nominal position of the second touch point in the tool axis of the working plane

Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)

Help graphic	Parameter
	Q1117 Diameter of 2nd position? Diameter of the second hole or the second stud Input: **0...9999.9999** or optional input: **"...-...+..."**: Evaluation of the tolerance, Page 1469
	Q1115 Geometry type (0-3)? Geometry of the objects: **0**: Position 1 = hole, and position 2 = hole **1**: Position 1 = stud, and position 2 = stud **2**: Position 1 = hole, and position 2 = stud **3**: Position 1 = stud, and position 2 = hole Input: **0**, **1**, **2**, **3**
	Q423 Number of probes? Number of touch points on the diameter Input: **3**, **4**, **5**, **6**, **7**, **8**
	**Q325 Starting angle?** Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: **-360.000...+360.000**
	Q1119 Arc angular length? Angular range in which the touch points are distributed. Input: **-359.999...+360.000**
	**Q320 Set-up clearance?** Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to clearance height. **0**: Move to clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary. Input: **0**, **1**, **2**
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: **0**: Retain the current position of the rotary axis. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. **2**: Automatically position the rotary axis without orienting the tool tip (**TURN**). Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point **2**: Correction based on the 2nd touch point **3**: Correction based on the averaged touch point position Input: **0**, **1**, **2**, **3**

Help graphic	Parameter
	Q1121 CONFIRM ROTATION? Define whether the control will use the determined misalignment as a basic rotation: **0**: No basic rotation **1**: Set basic rotation: The control transfers the misalignment to the preset table as basic transformations. **2**: Rotate the rotary table: The control transfers the misalignment to the preset table as offset. Input: **0**, **1**, **2**

Example

```
11 TCH PROBE 1411 PROBING TWO CIRCLES ~
    Q1100=+0                ;1ST POINT REF AXIS ~
    Q1101=+0                ;1ST POINT MINOR AXIS ~
    Q1102=+0                ;1ST POINT TOOL AXIS ~
    Q1116=+0                ;DIAMETER 1 ~
    Q1103=+0                ;2ND POINT REF AXIS ~
    Q1104=+0                ;2ND POINT MINOR AXIS ~
    Q1105=+0                ;2ND POINT TOOL AXIS ~
    Q1117=+0                ;DIAMETER 2 ~
    Q1115=+0                ;GEOMETRY TYPE ~
    Q423=+4                 ;NO. OF PROBE POINTS ~
    Q325=+0                 ;STARTING ANGLE ~
    Q1119=+360              ;ANGULAR LENGTH ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q1125=+2                ;CLEAR. HEIGHT MODE ~
    Q309=+0                 ;ERROR REACTION ~
    Q1126=+0                ;ALIGN ROTARY AXIS ~
    Q1120=+0                ;TRANSER POSITION ~
    Q1121=+0                ;CONFIRM ROTATION
```

29.2.6 Cycle 1412 INCLINED EDGE PROBING

Application

Touch probe cycle **1412** allows you to determine workpiece misalignment by probing two points on an inclined edge. The cycle determines the rotation based on the difference between the measured angle and the nominal angle.

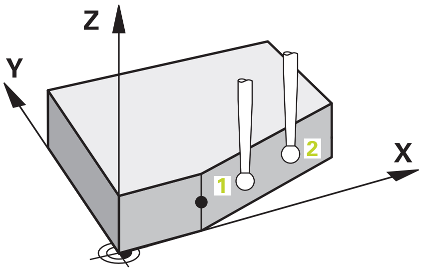
If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1676

Cycle **1412** additionally offers the following functions:

- If the probing position relative to the current datum is unknown, you can run the cycle in semi-automatic mode.
  Page 1463
- If you determine the actual position in advance, you can then transfer it to the cycle as the actual position.
  Page 1471

Cycle sequence



1. Following the positioning logic, the control positions the touch probe to the touch point **1** at rapid traverse **FMAX_PROBE**.
   Further information: "Positioning logic", Page 1456
2. The control then moves the touch probe to set-up clearance **Q320** at rapid traverse **FMAX_PROBE**. The sum of **Q320**, **SET_UP**, and the ball-tip radius is taken into account when probing in any probing direction.
3. Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate **F** from the touch probe table.
4. The control retracts the touch probe by the safety clearance in the direction opposite to the direction of probing.
5. If you programmed retraction to clearance height **Q1125**, the touch probe returns to clearance height.
6. The touch probe then moves to the touch point **2** and probes again.
7. Finally, the control retracts the touch probe to the clearance height (depending on **Q1125**) and saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to **Q955**	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to **Q982**	Measured deviations of touch point 1
Q983 to **Q985**	Measured deviations of touch point 2
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Mean value of all deviations from the ideal line of the first touch point
Q971	If you have programmed Cycle **1493 EXTRUSION PROBING** before: Mean value of all deviations from the ideal line of the second touch point

Notes

NOTICE

Danger of collision!

If you do not retract the touch probe to clearance height between two objects or touch points, there is danger of collision.

▶ Always move to clearance height between objects or touch points

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you program a tolerance in **Q1100**, **Q1101**, or **Q1102**, then this tolerance applies to the programmed nominal positions instead of to the touch points along the inclined edge. Use the parameter **TOLERANCE QS400** to program a tolerance for the surface normal along the inclined edge.

Note about rotary axes:

When determining the basic rotation in a tilted working plane, keep the following in mind:

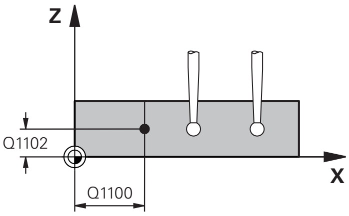
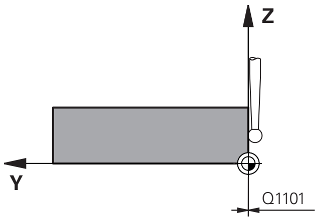
- If the current coordinates of the rotary axes and the defined tilt angles (3-D rotation window) match, the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
- If the current coordinates of the rotary axes and the defined tilt angles (3-D rotation window) do not match, the working plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS**, based on the tool axis.

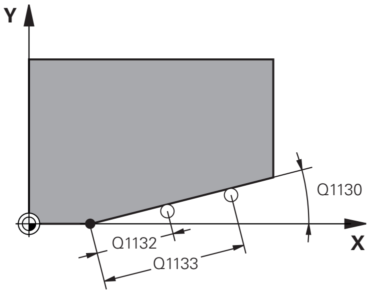
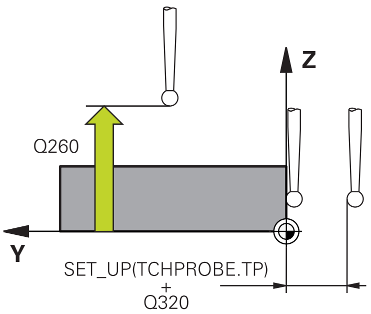
In the optional machine parameter **chkTiltingAxes** (no. 204601), the machine manufacturer defines a check verifying the tilting situation. If no checking is configured, the cycle always assumes that the working plane is consistent. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you need to adopt the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

Cycle parameters

Help graphic	Parameter
 	**Q1100 1st noml. position of ref. axis?** Absolute nominal position at which the inclined edge begins in the main axis. Input: **-99999.9999...+99999.9999** or optionally **?**, **+**, **-**, **@** ■ **?**: Semiautomatic mode, Page 1463 ■ **-, +**: Evaluation of the tolerance, Page 1469 ■ **@**: Transferring the actual position, Page 1471
	Q1101 1st noml. position of minor axis? Absolute nominal position at which the inclined edge begins in the secondary axis. Input: **-99999.9999...+99999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	QS400 Tolerance value? Tolerance band monitored by the cycle. The tolerance defines the deviation permitted for the surface normals along the inclined edge. This deviation is determined between the nominal coordinate and the actual coordinate of the workpiece. Examples: ■ **QS400 ="0.4-0.1"**: Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate –0.1. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate –0.1" ■ **QS400 =" "**: No tolerance band. ■ **QS400 ="0"**: No tolerance band. ■ **QS400 ="0.1+0.1"** : No tolerance band. Input: Max. **255** characters

Help graphic	Parameter
	**Q1130 Nominal angle for 1st line?** Nominal angle of the first straight line Input: **-180...+180**
	Q1131 Probing direction for 1st line? Probing direction for the first straight line: **+1**: The control rotates the probing direction by +90° with respect to the nominal angle **Q1130** **-1**: The control rotates the probing direction by –90° with respect to the nominal angle **Q1130** Input: **-1**, **+1**
	Q1132 First distance on 1st line? Distance between the beginning of the inclined edge and the first touch point. This value has an incremental effect. Input: **-999.999...+999.999**
	Q1133 Second distance on 1st line? Distance between the beginning of the inclined edge and the second touch point. This value has an incremental effect. Input: **-999.999...+999.999**
	Q1139 Plane for object (1-3)? Plane in which the control construes the nominal angle **Q1130** and the probing direction **Q1131**. **1**: The nominal angle is in the YZ plane. **2**: The nominal angle is in the ZX plane. **3**: The nominal angle is in the XY plane. Input: **1**, **2**, **3**
	**Q320 Set-up clearance?** Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to clearance height. **0**: Move to clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **1**: Move to clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**

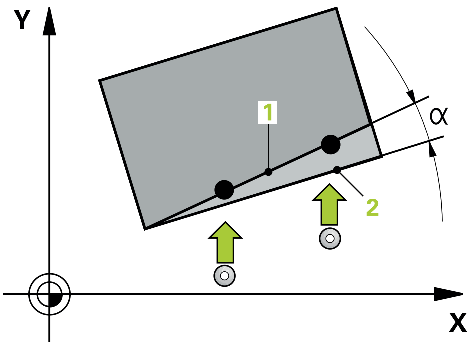
Help graphic	Parameter
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary. Input: **0**, **1**, **2**
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: **0**: Retain the current position of the rotary axis. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. **1**: Automatically position the rotary axis, and orient the tool tip (**MOVE**). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point **2**: Correction based on the 2nd touch point **3**: Correction based on the averaged touch point position Input: **0**, **1**, **2**, **3**
	Q1121 CONFIRM ROTATION? Define whether the control will use the determined misalignment as a basic rotation: **0**: No basic rotation **1**: Set basic rotation: The control transfers the misalignment to the preset table as basic transformations. **2**: Rotate the rotary table: The control transfers the misalignment to the preset table as offset. Input: **0**, **1**, **2**

Example

```
11 TCH PROBE 1412 INCLINED EDGE PROBING ~
    Q1100=+20               ;1ST POINT REF AXIS ~
    Q1101=+0                ;1ST POINT MINOR AXIS ~
    Q1102=-5                ;1ST POINT TOOL AXIS ~
    QS400="+0.1-0.1"        ;TOLERANCE ~
    Q1130=+30               ;NOMINAL ANGLE, 1ST LINE ~
    Q1131=+1                ;PROBE DIRECTION, 1ST LINE ~
    Q1132=+10               ;FIRST DISTANCE, 1ST LINE ~
    Q1133=+20               ;SECOND DISTANCE, 1ST LINE ~
    Q1139=+3                ;OBJECT PLANE ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q1125=+2                ;CLEAR. HEIGHT MODE ~
    Q309=+0                 ;ERROR REACTION ~
    Q1126=+0                ;ALIGN ROTARY AXIS ~
    Q1120=+0                ;TRANSER POSITION ~
    Q1121=+0                ;CONFIRM ROTATION
```

29.2.7 Touch probe cycles 4xx: fundamentals

Characteristics common to all touch probe cycles for measuring workpiece misalignment



In Cycles **400**, **401**, and **402**, you can use parameter **Q307 Preset value for rotation angle** to define whether the measurement result will be corrected by a known angle α (see figure). This enables you to measure the basic rotation against any straight line **1** of the workpiece and to establish the reference to the actual 0° direction **2**.

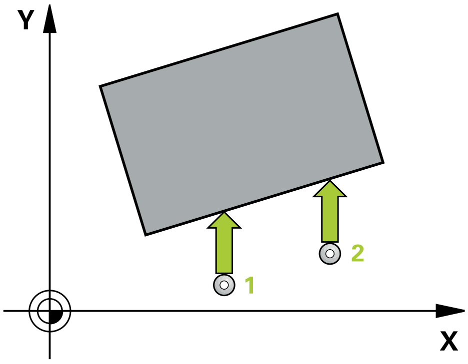
> These cycles do not work with 3-D ROT! In such a case, use Cycles **14xx**. Page 1462

29.2.8 Cycle 400 BASIC ROTATION

Application

Touch probe cycle **400** determines a workpiece misalignment by measuring two points, which must lie on a straight line. With the basic rotation function, the control compensates the measured value.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height and performs the basic rotation it determined.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

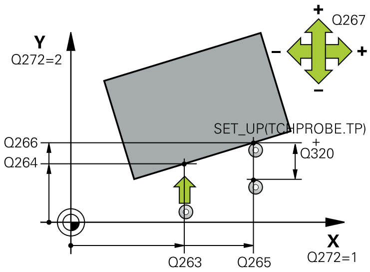
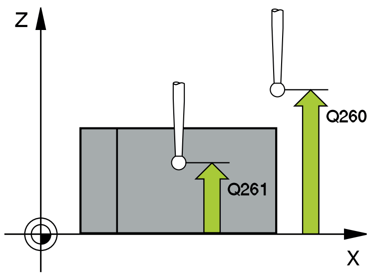
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q265 2nd measuring point in 1st axis? Coordinate of the second touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q266 2nd measuring point in 2nd axis? Coordinate of the second touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q272 Measuring axis (1=1st / 2=2nd)? Axis in the working plane in which the measurement will be performed: **1**: Main axis = measuring axis **2**: Secondary axis = measuring axis Input: **1**, **2**
	Q267 Trav. direction 1 (+1=+ / -1=-)? Direction in which the touch probe will approach the workpiece: **-1**: Negative traverse direction **+1**: Positive traverse direction Input: **-1**, **+1**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q307 Preset value for rotation angle If the misalignment will be measured against a straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. The value has an absolute effect. Input: **-360.000...+360.000**
	Q305 Preset number in table? Specify the number of the row in the preset table in which the control will save the calculated basic rotation. If you enter **Q305** = 0, the control automatically stores the calculated basic rotation in the ROT menu of the Manual Operation mode. Input: **0...99999**

Example

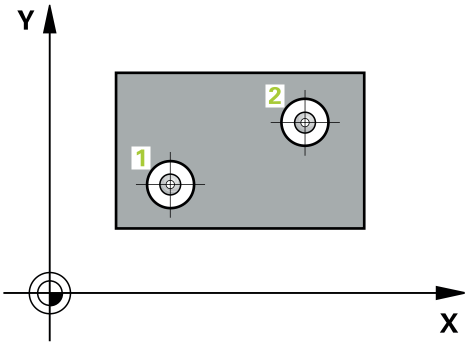
```
11 TCH PROBE 400 BASIC ROTATION ~
    Q263=+10            ;1ST POINT 1ST AXIS ~
    Q264=+3.5           ;1ST POINT 2ND AXIS ~
    Q265=+25            ;2ND POINT 1ST AXIS ~
    Q266=+2             ;2ND PNT IN 2ND AXIS ~
    Q272=+2             ;MEASURING AXIS ~
    Q267=+1             ;TRAVERSE DIRECTION ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q301=+0             ;MOVE TO CLEARANCE ~
    Q307=+0             ;PRESET ROTATION ANG. ~
    Q305=+0             ;NUMBER IN TABLE
```

29.2.9 Cycle 401 ROT OF 2 HOLES

Application

Touch probe cycle **401** measures the center points of two holes. The control then calculates the angle between the main axis of the working plane and the line connecting the hole center points. With the basic rotation function, the control compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed center point of the first hole **1**.

   Further information: "Positioning logic", Page 1456
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

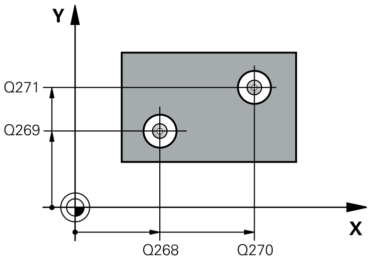
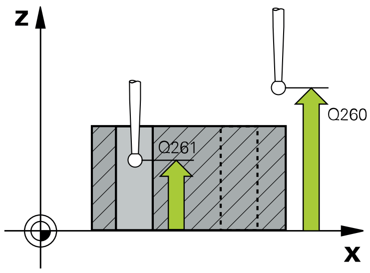
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:
  - C for tool axis Z
  - B for tool axis Y
  - A for tool axis X

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q268 1st hole: center in 1st axis?** Center of the first hole in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q269 1st hole: center in 2nd axis? Center of the first hole in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q270 2nd hole: center in 1st axis? Center of the second hole in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q271 2nd hole: center in 2nd axis? Center of the second hole in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q307 Preset value for rotation angle If the misalignment will be measured against a straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. The value has an absolute effect. Input: **-360.000...+360.000**

Help graphic	Parameter
	Q305 Number in table? Enter the number of a row in the preset table. The control will make the corresponding entry in the following row: **Q305** = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0. **Q305** > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**). **Q305 depends on the following parameters:** ■ **Q337** = 0 and, at the same time, **Q402** = 0: A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, the basic rotation is entered in the **SPC** column). ■ **Q337** = 0 and, at the same time, **Q402** = 1: The parameter **Q305** is not effective. ■ **Q337** = 1: The parameter **Q305** has the effect described above. Input: **0...99999**
	Q402 Basic rotation/alignment (0/1) Define whether the control will set the determined misalignment as a basic rotation or will compensate it by rotating the rotary table: **0**: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**) **1**: Rotate the rotary table: An entry will be made in the corresponding **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column); in addition, the corresponding axis will be rotated Input: **0**, **1**
	Q337 Set to zero after alignment? Define whether the control will set the position display of the corresponding rotary axis to 0 after the alignment: **0**: The position display is not set to 0 after the alignment **1**: After the alignment, the position display is set to 0, provided you have defined **Q402 = 1** Input: **0**, **1**

Example

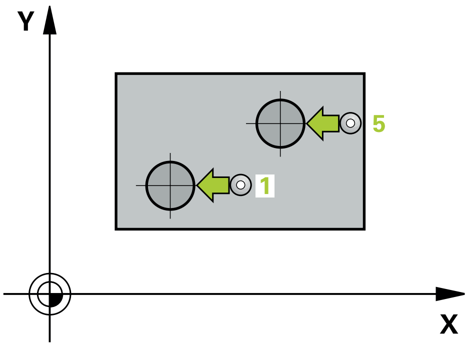
```
11 TCH PROBE 401 ROT OF 2 HOLES ~
    Q268=-37                ;1ST CENTER 1ST AXIS ~
    Q269=+12                ;1ST CENTER 2ND AXIS ~
    Q270=+75                ;2ND CENTER 1ST AXIS ~
    Q271=+20                ;2ND CENTER 2ND AXIS ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q260=+20                ;CLEARANCE HEIGHT ~
    Q307=+0                 ;PRESET ROTATION ANG. ~
    Q305=+0                 ;NUMBER IN TABLE ~
    Q402=+0                 ;COMPENSATION ~
    Q337=+0                 ;SET TO ZERO
```

29.2.10 Cycle 402 ROT OF 2 STUDS

Application

Touch probe cycle **402** measures the center points of two cylindrical studs. The control then calculates the angle between the main axis of the working plane and the line connecting the stud center points. With the basic rotation function, the control compensates the calculated value. As an alternative, you can also compensate the determined misalignment by rotating the rotary table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from FMAX column) to touch point **1** of the first stud.

   Further information: "Positioning logic", Page 1456

2 Then the touch probe moves to the entered **measuring height 1** and probes four points to find the center of the first stud. The touch probe moves along a circular arc between the touch points, each of which is offset by 90°.
3 The touch probe returns to the clearance height and then moves to the touch point **5** of the second stud.
4 The control moves the touch probe to the entered **measuring height 2** and probes four points to determine the center of the second stud.
5 Then the control returns the touch probe to the clearance height and performs the calculated basic rotation.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

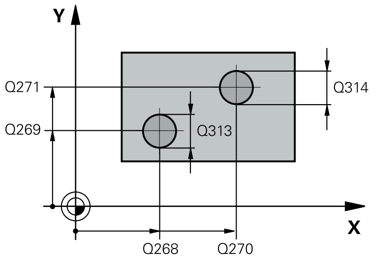
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:
  - C for tool axis Z
  - B for tool axis Y
  - A for tool axis X

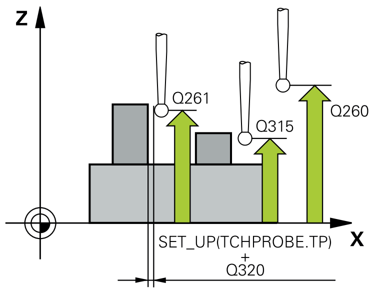
Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic





Parameter

Q268 1st stud: center in 1st axis?
Center of the first stud in the main axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q269 1st stud: center in 2nd axis?
Center of the first stud in the secondary axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q313 Diameter of stud 1?
Approximate diameter of the first stud. Enter a value that is more likely to be too large than too small.
Input: **0...99999.9999**

Q261 Meas. height stud 1 in TS axis?
Coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 1 will be measured. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q270 2nd stud: center in 1st axis?
Center of the second stud in the main axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q271 2nd stud: center in 2nd axis?
Center of the second stud in the secondary axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q314 Diameter of stud 2?
Approximate diameter of the second stud. Enter a value that is more likely to be too large than too small.
Input: **0...99999.9999**

Q315 Meas. height stud 2 in TS axis?
Coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 2 will be measured. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?
Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.
Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?
Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.
Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q307 Preset value for rotation angle If the misalignment will be measured against a straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. The value has an absolute effect. Input: **-360.000...+360.000**
	Q305 Number in table? Enter the number of a row in the preset table. The control will make the corresponding entry in the following row: **Q305** = 0: The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0. **Q305** > 0: The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C_OFFS**). **Q305 depends on the following parameters:** ■ **Q337** = 0 and, at the same time, **Q402** = 0: A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, the basic rotation is entered in the **SPC** column). ■ **Q337** = 0 and, at the same time, **Q402** = 1: The parameter **Q305** is not effective. ■ **Q337** = 1: The parameter **Q305** has the effect described above. Input: **0...99999**

Help graphic	Parameter
	Q402 Basic rotation/alignment (0/1) Define whether the control will set the determined misalignment as a basic rotation or will compensate it by rotating the rotary table: **0**: Set basic rotation: The control saves the basic rotation (example: for tool axis Z, the control uses column **SPC**) **1**: Rotate the rotary table: An entry will be made in the corresponding **Offset** column of the preset table (example: for tool axis Z, the control uses the **C_OFFS** column); in addition, the corresponding axis will be rotated Input: **0**, **1**
	Q337 Set to zero after alignment? Define whether the control will set the position display of the corresponding rotary axis to 0 after the alignment: **0**: The position display is not set to 0 after the alignment **1**: After the alignment, the position display is set to 0, provided you have defined **Q402 = 1** Input: **0**, **1**

Example

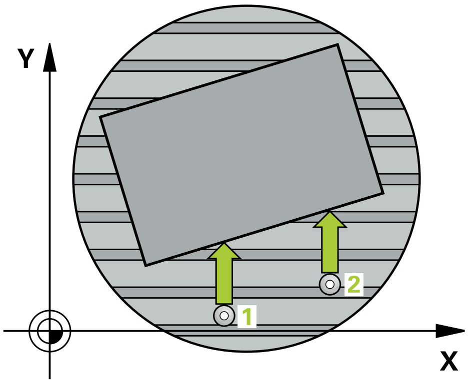
```
11 TCH PROBE 402 ROT OF 2 STUDS ~
    Q268=-37              ;1ST CENTER 1ST AXIS ~
    Q269=+12              ;1ST CENTER 2ND AXIS ~
    Q313=+60              ;DIAMETER OF STUD 1 ~
    Q261=-5               ;MEAS. HEIGHT STUD 1 ~
    Q270=+75              ;2ND CENTER 1ST AXIS ~
    Q271=+20              ;2ND CENTER 2ND AXIS ~
    Q314=+60              ;DIAMETER OF STUD 2 ~
    Q315=-5               ;MEAS. HEIGHT STUD 2 ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q260=+20              ;CLEARANCE HEIGHT ~
    Q301=+0               ;MOVE TO CLEARANCE ~
    Q307=+0               ;PRESET ROTATION ANG. ~
    Q305=+0               ;NUMBER IN TABLE ~
    Q402=+0               ;COMPENSATION ~
    Q337=+0               ;SET TO ZERO
```

29.2.11 Cycle 403 ROT IN ROTARY AXIS

Application

Touch probe cycle **403** determines a workpiece misalignment by measuring two points, which must lie on a straight line. The control compensates the determined misalignment by rotating the A, B, or C axis. The workpiece can be clamped in any position on the rotary table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height and rotates the rotary axis, which was defined in the cycle, by the measured value. Optionally, you can specify whether the control is to set the determined rotation angle to 0 in the preset table or in the datum table.

Notes

NOTICE

Danger of collision!

If the control positions the rotary axis automatically, a collision might occur.

- Check for possible collisions between the tool and any elements positioned on the table
- Select the clearance height to prevent collisions

NOTICE

Danger of collision!

If you set parameter **Q312** Axis for compensating movement? to 0, then the cycle will automatically determine the rotary axis to be aligned (recommended setting). When doing so, it determines an angle that depends on the sequence of the touch points. The measured angle goes from the first to the second touch point. If you select the A, B or C axis as compensation axis in parameter **Q312**, the cycle determines the angle, regardless of the sequence of the probing points. The calculated angle lies in the range from –90° to +90°.

- After alignment, check the position of the rotary axis.

NOTICE

Danger of collision!

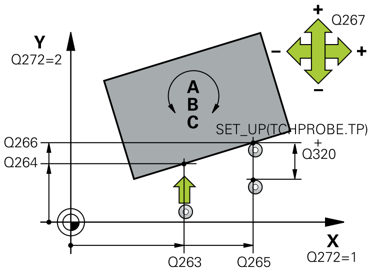
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic | **Parameter**



Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

1: Main axis = measuring axis

2: Secondary axis = measuring axis

3: Touch probe axis = measuring axis

Input: **1**, **2**, **3**

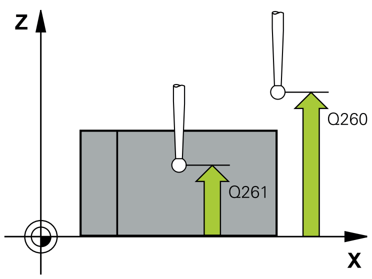
Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

-1: Negative traverse direction

+1: Positive traverse direction

Input: **-1**, **+1**



Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q312 Axis for compensating movement? Define the rotary axis in which the control will compensate the measured misalignment: **0**: Automatic mode – the control uses the active kinematics to determine the rotary axis to be aligned. In Automatic mode the first rotary axis of the table (as viewed from the workpiece) is used as compensation axis. This is the recommended setting! **4**: Compensate misalignment with rotary axis A **5**: Compensate misalignment with rotary axis B **6**: Compensate misalignment with rotary axis C Input: **0**, **4**, **5**, **6**
	Q337 Set to zero after alignment? Define whether the control will set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment. **0**: Do not set the angle of the rotary axis to 0 in the table after the alignment **1**: Set the angle of the rotary axis to 0 in the table after the alignment Input: **0**, **1**

Help graphic	Parameter
	Q305 Number in table? Specify the number of the row in the preset table in which the control will enter the basic rotation. **Q305** = 0: The rotary axis is zeroed in row number 0 of the preset table. The control will make an entry in the **OFFSET** column. In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0. **Q305** > 0: Specify the number of the row in the preset table in which the control will zero the rotary axis. The control will make an entry in the **OFFSET** column of the preset table. **Q305 depends on the following parameters:** ■ **Q337 = 0**: Parameter **Q305** is not effective ■ **Q337 = 1**: Parameter **Q305** has the effect described above ■ **Q312 = 0**: Parameter **Q305** has the effect described above ■ **Q305 Number in table? Q312 > 0**: The entry in **Q305** is ignored. The control will make an entry in the **OFFSET** column, in the row of the preset table that was active when the cycle was called. Input: **0...99999**

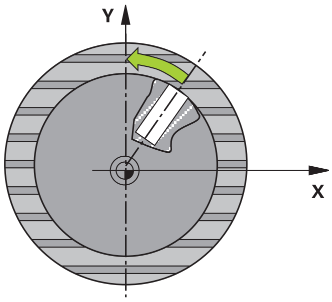
Help graphic	Parameter
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **0**: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **0**, **1**
	Q380 Ref. angle in ref. axis? Angle to which the control will align the probed straight line. Only effective if the rotary axis is in automatic mode or if C is selected (**Q312** = 0 or 6). Input: **0...360**

Example

```
11 TCH PROBE 403 ROT IN ROTARY AXIS ~
    Q263=+0             ;1ST POINT 1ST AXIS ~
    Q264=+0             ;1ST POINT 2ND AXIS ~
    Q265=+20            ;2ND PNT IN 1ST AXIS ~
    Q266=+30            ;2ND POINT 2ND AXIS ~
    Q272=+1             ;MEASURING AXIS ~
    Q267=-1             ;TRAVERSE DIRECTION ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q301=+0             ;MOVE TO CLEARANCE ~
    Q312=+0             ;COMPENSATION AXIS ~
    Q337=+0             ;SET TO ZERO ~
    Q305=+1             ;NUMBER IN TABLE ~
    Q303=+1             ;MEAS. VALUE TRANSFER ~
    Q380=+90            ;REFERENCE ANGLE
```

29.2.12 Cycle 405 ROT IN C AXIS

Application

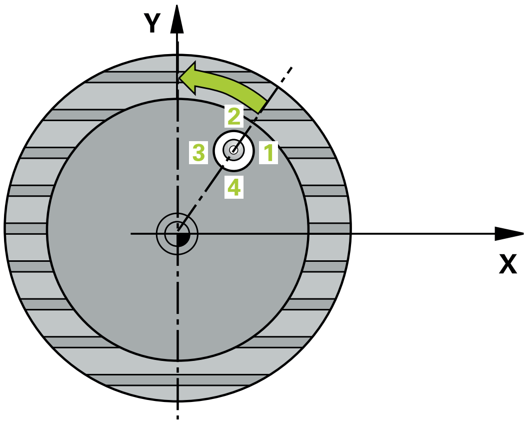


With touch probe cycle **405**, you can measure

- the angular offset between the positive Y axis of the active coordinate system and the center line of a hole
- the angular offset between the nominal position and the actual position of a hole center point

The control compensates the determined angular offset by rotating the C axis. The workpiece can be clamped in any position on the rotary table, but the Y coordinate of the hole must be positive. If you measure the angular misalignment of the hole with touch probe axis Y (horizontal position of the hole), it may be necessary to execute the cycle more than once because the measuring strategy causes an inaccuracy of approx. 1% of the misalignment.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.

3 Then, the touch probe moves along a circular arc, either at measuring height or at clearance height, to the next touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times and then positions the touch probe on the calculated hole center.

5 Finally, the control returns the touch probe to the clearance height and aligns the workpiece by rotating the rotary table. The control rotates the rotary table in such a way that the hole center, after compensation, lies in the direction of the positive Y axis or at the nominal position of the hole center point—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in the parameter **Q150**.

Notes

NOTICE

Danger of collision!

If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.

- Make sure that there is no material in the pocket/hole
- To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

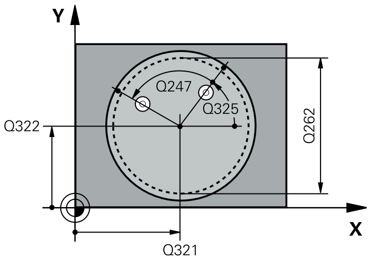
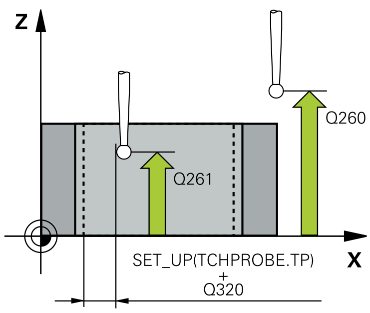
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- The smaller the stepping angle, the less accurately the control can calculate the circle center point. Minimum input value: 5°.

Cycle parameters

Help graphic	Parameter
	**Q321 Center in 1st axis?** Center of the hole in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q322 Center in 2nd axis? Center of the hole in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point with the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point with the nominal position (angle resulting from the position of the hole center). The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input: **0...99999.9999**
	Q325 Starting angle? Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect. Input: **-120...+120**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q337 Set to zero after alignment? **0**: Set the display of the C axis to 0 and write to **C_Offset** of the active row of the datum table **> 0**: Write the measured angular offset to the datum table. Row number = value in **Q337**. If a C-axis shift is registered in the datum table, the control adds the measured angular offset with the correct sign, positive or negative. Input: **0...2999**

Example

```
11 TCH PROBE 405 ROT IN C AXIS ~
    Q321=+50            ;CENTER IN 1ST AXIS ~
    Q322=+50            ;CENTER IN 2ND AXIS ~
    Q262=+10            ;NOMINAL DIAMETER ~
    Q325=+0             ;STARTING ANGLE ~
    Q247=+90            ;STEPPING ANGLE ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q301=+0             ;MOVE TO CLEARANCE ~
    Q337=+0             ;SET TO ZERO
```

29.2.13 Cycle 404 SET BASIC ROTATION

Application

With touch probe cycle **404**, you can set any basic rotation automatically during program run or save it to the preset table. You can also use Cycle **404** if you want to reset an active basic rotation.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

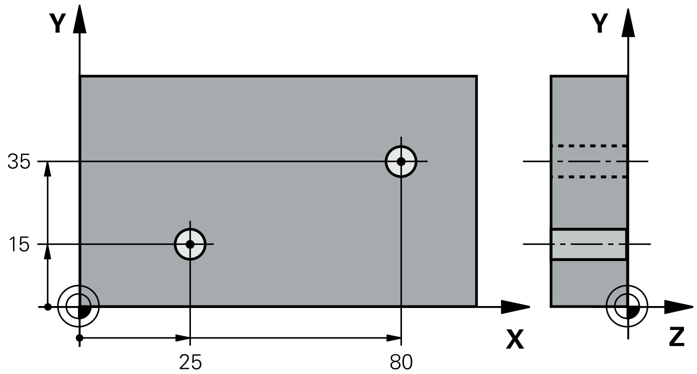
Cycle parameters

Help graphic	Parameter
	Q307 Preset value for rotation angle Angle value at which the basic rotation will be set. Input: **-360.000...+360.000**
	Q305 Preset number in table?: Specify the number of the row in the preset table in which the control will save the calculated basic rotation. If you enter **Q305** = 0 or **Q305** = –1, the control additionally saves the calculated basic rotation in the basic rotation menu (**Probing rot**) of **Manual Operation** mode. **-1**: Overwrite and activate the active preset **0**: Copy the active preset to row 0 of the preset table, write the basic rotation to row 0 of the preset table, and activate preset 0 **> 1**: Save the basic rotation to the specified preset. The preset is not activated. Input: **-1...99999**

Example

```
11 TCH PROBE 404 SET BASIC ROTATION ~
    Q307=+0                 ;PRESET ROTATION ANG. ~
    Q305=-1                 ;NUMBER IN TABLE
```

29.2.14 Example: Determining a basic rotation from two holes



- **Q268** = Center of the 1st hole: X coordinate
- **Q269** = Center of the 1st hole: Y coordinate
- **Q270** = Center of the 2nd hole: X coordinate
- **Q271** = Center of the 2nd hole: Y coordinate
- **Q261** = Coordinate in the touch probe axis in which the measurement is performed
- **Q307** = Angle of the reference line
- **Q402** = Compensation of workpiece misalignment by rotating the table
- **Q337** = Set the display to zero after the alignment

0 BEGIN PGM TOUCHPROBE MM	
1 TOOL CALL 600 Z	
2 TCH PROBE 401 ROT OF 2 HOLES ~	
Q268=+25 ;1ST CENTER 1ST AXIS ~	
Q269=+15 ;1ST CENTER 2ND AXIS ~	
Q270=+80 ;2ND CENTER 1ST AXIS ~	
Q271=+35 ;2ND CENTER 2ND AXIS ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q307=+0 ;PRESET ROTATION ANG. ~	
Q305=+0 ;NUMBER IN TABLE	
Q402=+1 ;COMPENSATION ~	
Q337=+1 ;SET TO ZERO	
3 CALL PGM 35	; Call the part program
4 END PGM TOUCHPROBE MM	

29.3 Touch Probe Cycles: Automatic Preset Measurement

29.3.1 Overview

The control offers cycles for automatic preset measurement.

> The control must be specifically prepared by the machine manufacturer for the use of a touch probe.
> HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.

Cycle		Call	Further information
1400	**POSITION PROBING** ■ Measurement of single position ■ Definition of preset, if necessary	**DEF-**active	Page 1525
1401	**CIRCLE PROBING** ■ Measurement of points on the inside or outside of a circle ■ Definition of circle center as preset, if necessary	**DEF-**active	Page 1529
1402	**SPHERE PROBING** ■ Measurement of points on a sphere ■ Definition of sphere center as preset, if necessary	**DEF-**active	Page 1533
410	**PRESET INSIDE RECTAN** ■ Measurement of inside length and width of a rectangle ■ Definition of rectangle center as preset	**DEF-**active	Page 1540
411	**PRESET OUTS. RECTAN** ■ Measurement of outside length and width of a rectangle ■ Definition of rectangle center as preset	**DEF-**active	Page 1545
412	**PRESET INSIDE CIRCLE** ■ Measurement of any four points on the inside of a circle ■ Definition of circle center as preset	**DEF-**active	Page 1551
413	**PRESET OUTS. CIRCLE** ■ Measurement of any four points on the outside of a circle ■ Definition of circle center as preset	**DEF-**active	Page 1557
414	**PRESET OUTS. CORNER** ■ Measurement of two straight lines on the outside ■ Definition of intersection of the lines as preset	**DEF-**active	Page 1563
415	**PRESET INSIDE CORNER** ■ Measurement of two straight lines on the inside ■ Definition of intersection of the lines as preset	**DEF-**active	Page 1568
416	**PRESET CIRCLE CENTER**	**DEF-**active	Page 1574

Cycle		Call	Further information
	■ Measurement of any three holes on a circular hole pattern ■ Definition of center of circular hole pattern as preset		
417	**PRESET IN TS AXIS** ■ Measurement of any position in the tool axis ■ Definition of any position as preset	**DEF**-active	Page 1581
418	**PRESET FROM 4 HOLES** ■ Measurement of two holes on each line crosswise ■ Definition of the intersection of the lines as preset	**DEF**-active	Page 1585
419	**PRESET IN ONE AXIS** ■ Measurement of any position in a selectable axis ■ Definition of any position in a selectable axis as preset	**DEF**-active	Page 1590
408	**SLOT CENTER PRESET** ■ Measurement of width of an inside slot ■ Definition of slot center as preset	**DEF**-active	Page 1593
409	**RIDGE CENTER PRESET** ■ Measurement of width of an outside ridge ■ Definition of ridge center as preset	**DEF**-active	Page 1598

29.3.2 Fundamentals of touch probe cycles 14xx for presetting

Characteristics common to all touch probe cycles 14xx for preset setting

Preset and tool axis

The control sets the preset in the working plane based on the touch probe axis that you defined in your measuring program.

Active touch probe axis	Preset setting in
Z	X and Y
Y	Z and X
X	Y and Z

Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q9xx**. You can use the parameters in your NC program. Note the table of result parameters listed with every cycle description.

29.3.3 Cycle 1400 POSITION PROBING

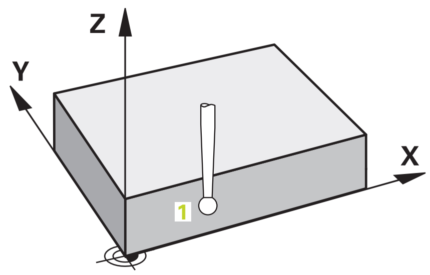
Application

Touch probe cycle **1400** measures any position in a selectable axis. You can apply the result to the active row of the preset table.

If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1676

Cycle sequence



1. Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control takes into account the set-up clearance **Q320** during pre-positioning.
   Further information: "Positioning logic", Page 1456
2. Then the touch probe moves to the entered measuring height and measures the actual position with a single probing movement.
3. The control returns the touch probe to the clearance height.
4. The control saves the measured position in the following Q parameters. If **Q1120 = 1**, then the control writes the measured position to the active row of the preset table.
   Further information: "Fundamentals of touch probe cycles 14xx for presetting", Page 1525

Q parameter number	Meaning
Q950 to **Q952**	Measured position 1 in the main axis, secondary axis, and tool axis
Q980 to **Q982**	Measured deviations of touch point 1
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Mean value of all deviations from the ideal line of the second touch point

Notes

NOTICE

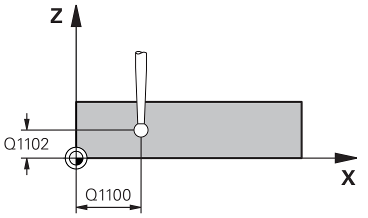
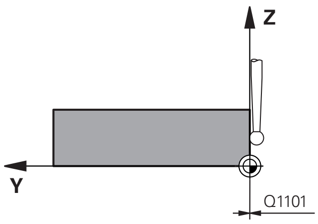
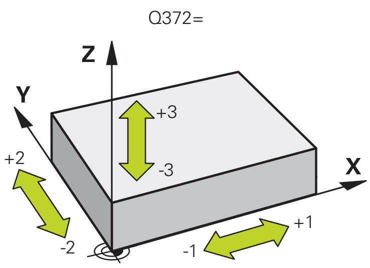
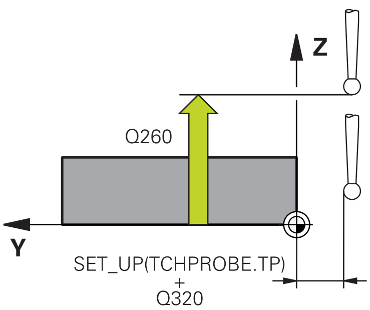
Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the first touch point in the main axis of the working plane Input: **-99999.9999...+99999.9999** or optionally **?**, **-**, **+**, **@** **?**: Semiautomatic mode, Page 1463 **-**, **+**: Evaluation of the tolerance, Page 1469 **@**: Transferring the actual position, Page 1471
	**Q1101 1st noml. position of minor axis?** Absolute nominal position of the first touch point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	**Q372 Probe direction (-3 to +3)?** Axis defining the direction of probing. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis. Input: **-3**, **-2**, **-1**, **+1**, **+2**, **+3**
	**Q320 Set-up clearance?** Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q1125 Traverse to clearance height? Positioning behavior between the touch points: **-1**: Do not move to clearance height. **0, 1, 2**: Move to clearance height before and after the touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point Input: **0**, **1**

Example

```
11 TCH PROBE 1400 POSITION PROBING ~
    Q1100=+25              ;1ST POINT REF AXIS ~
    Q1101=+25              ;1ST POINT MINOR AXIS ~
    Q1102=-5               ;1ST POINT TOOL AXIS ~
    Q372=+0                ;PROBING DIRECTION ~
    Q320=+0                ;SET-UP CLEARANCE ~
    Q260=+50               ;CLEARANCE HEIGHT ~
    Q1125=+1               ;CLEAR. HEIGHT MODE ~
    Q309=+0                ;ERROR REACTION ~
    Q1120=+0               ;TRANSER POSITION
```

29.3.4 Cycle 1401 CIRCLE PROBING

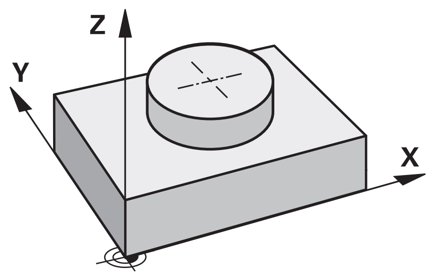
Application

Touch probe cycle **1401** determines the center point of a circular pocket or circular stud. You can transfer the result to the active row of the preset table.

If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 1676

Cycle sequence



1. Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed touch point. The control takes the set-up clearance **Q320** into account during pre-positioning.

   Further information: "Positioning logic", Page 1456
2. Then the touch probe moves to the entered measuring height **Q1102** and measures the actual position of the first touch point.
3. The control returns the touch probe to the clearance height **Q260** at **FMAX_PROBE** and then moves it to the next touch point.
4. The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
5. Depending on the definition of **Q423 NO. OF PROBE POINTS**, the control repeats the steps 3 to 4.
6. The control returns the touch probe to the clearance height **Q260**.
7. The control saves the measured position in the following Q parameters. If **Q1120 = 1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx for presetting", Page 1525

Q parameter number	Meaning
Q950 to **Q952**	Measured circle center point in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to **Q982**	Measured deviations of the circle center point
Q996	Measured deviation of the diameter
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap
Q970	If you have programmed Cycle **1493 EXTRUSION PROBING**: Mean value of all deviations from the ideal line of the first circle center point
Q973	If you have programmed Cycle **1493 EXTRUSION PROBING**: Mean value of all deviations of the diameters of circle 1

Notes

NOTICE

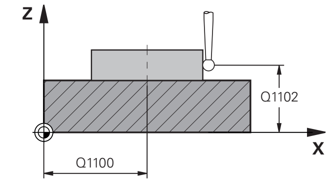
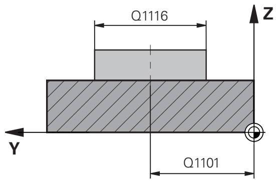
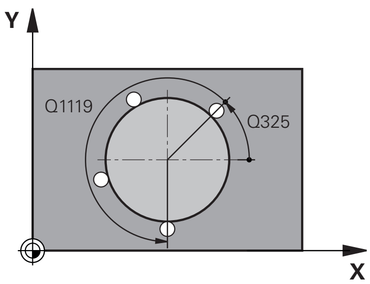
Danger of collision!

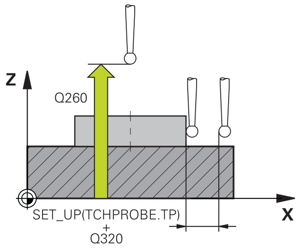
When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the center point in the main axis of the working plane. Input: **-99999.9999...+99999.9999** or optional input: **"?..."**: Semiautomatic mode, Page 1463 **"...-...+..."**: Evaluation of the tolerance, Page 1469 **"...@..."**: Transferring the actual position, Page 1471
	Q1101 1st noml. position of minor axis? Absolute nominal position of the center point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1116 Diameter of 1st position? Diameter of the first hole or the first stud **"...-...+..."**: Evaluation of the tolerance, Page 1469 Input: **0...9999.9999** or optional input:
	Q1115 Geometry type (0/1)? Geometry of the object: **0**: Hole **1**: Stud Input: **0**, **1**
	Q423 Number of probes? Number of touch points on the diameter Input: **3**, **4**, **5**, **6**, **7**, **8**
	**Q325 Starting angle?** Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: **-360.000...+360.000**
	Q1119 Arc angular length? Angular range in which the touch points are distributed. Input: **-359.999...+360.000**

Help graphic	Parameter
	**Q320 Set-up clearance?** Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points **-1**: Do not move to clearance height. **0**, **1**: Move to clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on the 1st touch point Input: **0**, **1**

Example

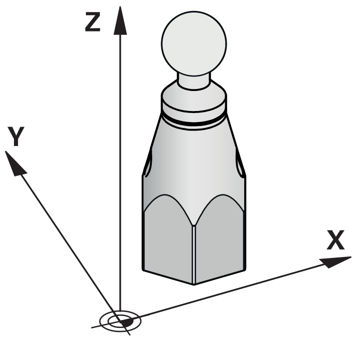
```
11 TCH PROBE 1401 CIRCLE PROBING ~
    Q1100=+25              ;1ST POINT REF AXIS ~
    Q1101=+25              ;1ST POINT MINOR AXIS ~
    Q1102=-5               ;1ST POINT TOOL AXIS ~
    QS1116=+10             ;DIAMETER 1 ~
    Q1115=+0               ;GEOMETRY TYPE ~
    Q423=+3                ;NO. OF PROBE POINTS ~
    Q325=+0                ;STARTING ANGLE ~
    Q1119=+360             ;ANGULAR LENGTH ~
    Q320=+0                ;SET-UP CLEARANCE ~
    Q260=+50               ;CLEARANCE HEIGHT ~
    Q1125=+1               ;CLEAR. HEIGHT MODE ~
    Q309=+0                ;ERROR REACTION ~
    Q1120=+0               ;TRANSER POSITION
```

29.3.5 Cycle 1402 SPHERE PROBING

Application

Touch probe cycle **1402** determines the center point of a sphere. You can apply the result to the active row of the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed touch point. The control takes the set-up clearance **Q320** into account during pre-positioning.

   Further information: "Positioning logic", Page 1456
2 Then the touch probe moves to the entered measuring height **Q1102** and measures the actual position of the first touch point with a single probing movement.
3 The control returns the touch probe to the clearance height **Q260** at **FMAX_PROBE** and then moves it to the next touch point.
4 The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
5 Depending on the definition of **Q423** Number of Probes, the control repeats the steps 3 to 4.
6 The control moves the touch probe in the tool axis by the set-up clearance to a position above the sphere.
7 The touch probe moves to the center of the sphere and probes another touch point.
8 The touch probe returns to the clearance height **Q260**.
9 The control saves the measured position in the following Q parameters. If **Q1120 = 1**, then the control writes the measured position to the active row of the preset table.

   Further information: "Fundamentals of touch probe cycles 14xx for presetting", Page 1525

Q parameter number	Meaning
Q950 to **Q952**	Measured circle center point in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to **Q982**	Measured deviations of the circle center point
Q996	Measured deviations of the diameter
Q183	Workpiece status ■ **-1** = Not defined ■ **0** = Good ■ **1** = Rework ■ **2** = Scrap

Notes

NOTICE

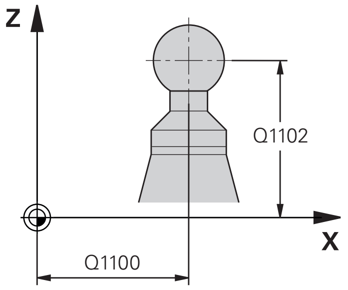
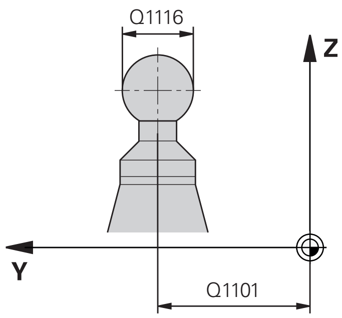
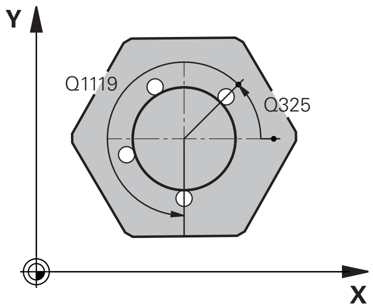
Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you have programmed Cycle **1493 EXTRUSION PROBING** before, the control will ignore it during the execution of Cycle **1402 SPHERE PROBING**.

Cycle parameters

Help graphic	Parameter
	**Q1100 1st noml. position of ref. axis?** Absolute nominal position of the center point in the main axis of the working plane. Input: **-99999.9999...+99999.9999** or optional input: **"?..."**: Semiautomatic mode, Page 1463 **"...-...+..."**: Evaluation of the tolerance, Page 1469 **"...@..."**: Transferring the actual position, Page 1471
	**Q1101 1st noml. position of minor axis?** Absolute nominal position of the center point in the secondary axis of the working plane Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: **-99999.9999...+9999.9999** or optional input (see **Q1100**)
	Q1116 Diameter of 1st position? Diameter of the sphere **"...-...+..."**: Evaluation of the tolerance, Page 1469 Input: **0...9999.9999** or optional input (see **Q1100**)
	Q423 Number of probes? Number of touch points on the diameter Input: **3**, **4**, **5**, **6**, **7**, **8**
	**Q325 Starting angle?** Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: **-360.000...+360.000**
	Q1119 Arc angular length? Angular range in which the touch points are distributed. Input: **-359.999...+360.000**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q1125 Traverse to clearance height? Positioning behavior between the touch points **-1**: Do not move to clearance height. **0**, **1**: Move to clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**. **2**: Move to clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**. Input: **-1**, **0**, **+1**, **+2**
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: **0**: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. **1**: Interrupt program run when tolerance is exceeded. The control opens a window with the results. **2**: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary. Input: **0**, **1**, **2**
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: **0**: No correction **1**: Correction based on center of the sphere Input: **0**, **1**

Example

```
11 TCH PROBE 1402 SPHERE PROBING ~
    Q1100=+25             ;1ST POINT REF AXIS ~
    Q1101=+25             ;1ST POINT MINOR AXIS ~
    Q1102=-5              ;1ST POINT TOOL AXIS ~
    QS1116=+10            ;DIAMETER 1 ~
    Q423=+3               ;NO. OF PROBE POINTS ~
    Q325=+0               ;STARTING ANGLE ~
    Q1119=+360            ;ANGULAR LENGTH ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q260=+50              ;CLEARANCE HEIGHT ~
    Q1125=+1              ;CLEAR. HEIGHT MODE ~
    Q309=+0               ;ERROR REACTION ~
    Q1120=+0              ;TRANSER POSITION
```

29.3.6 Fundamentals of touch probe cycles 4xx for preset setting

Characteristics common to all touch probe cycles 4xx for preset setting

Depending on the setting of the optional **CfgPresetSettings** machine parameter (no. 204600), the control will check during probing whether the position of the rotary axis matches the tilting angles **3-D ROTATION**. If that is not the case, the control displays an error message.

The control offers cycles for automatically determining presets and handling them as follows:

- Setting the calculated values directly as display values
- Writing the calculated values to the preset table
- Writing the calculated values to a datum table

Preset and touch probe axis

The control determines the preset in the working plane based on the touch probe axis that you defined in your measuring program.

Active touch probe axis	Set preset in
Z	X and Y
Y	Z and X
X	Y and Z

Saving the calculated preset

In all cycles for presetting, you can use input parameters **Q303** and **Q305** to define how the control is to save the calculated preset:

- **Q305 = 0, Q303 = 1:**
  The control copies the active preset to row 0, changes it and activates row 0, deleting simple transformations.
- **Q305 not equal to 0, Q303 = 0:**
  The result is written to the datum table, row **Q305**; **activate the datum with TRANS DATUM in the NC program**

  Further information: "Datum shift with TRANS DATUM", Page 972
- **Q305 not equal to 0, Q303 = 1:**
  The result is written to the preset table, row **Q305**; **use Cycle 247 to activate the preset in the NC program**
- **Q305 not equal to 0, Q303 = -1**

> This combination can only occur if you
> - read in NC programs (containing Cycles **410** to **418**) that were created on a TNC 4xx
> - read in NC programs (containing Cycles **410** to **418**) that were created with an older software version of an iTNC 530
> - did not specifically define the measured-value transfer with parameter **Q303** when defining the cycle
>
> In these cases, the control outputs an error message, since the complete handling of REF-referenced datum tables has changed. You must define a measured-value transfer yourself with parameter **Q303**.

Measurement results in Q parameters

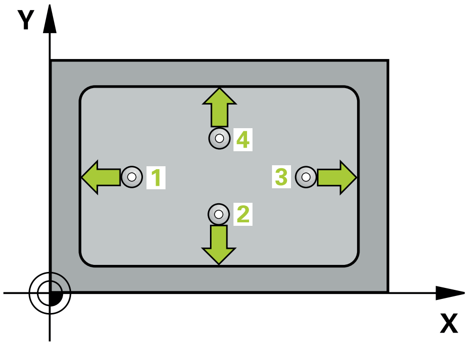
The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. You can use these parameters in your NC program. Note the table of result parameters listed with every cycle description.

29.3.7 Cycle 410 PRESET INSIDE RECTAN

Application

Touch probe cycle **410** finds the center of a rectangular pocket and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 The control returns the touch probe to the clearance height.
6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
7 Then the control saves the actual values in the Q parameters listed below.
8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

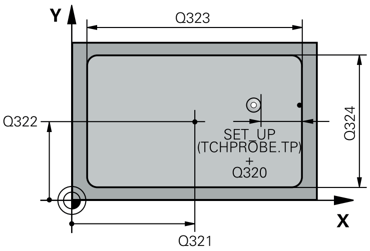
To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides. If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.

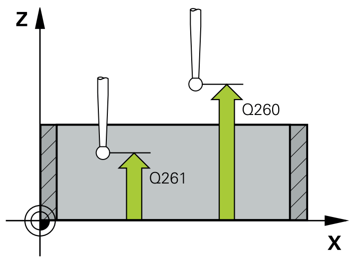
- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic





Parameter

Q321 Center in 1st axis?

Center of the pocket in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q322 Center in 2nd axis?

Center of the pocket in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q323 First side length?

Pocket length, parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q324 Second side length?

Pocket length, parallel to the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q261 Measuring height in probe axis?

The value has an absolute effect. Input: –99999.9999... +99999.9999

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0**, **1**

Help graphic	Parameter
	Q305 Number in table? Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**

Help graphic	Parameter
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

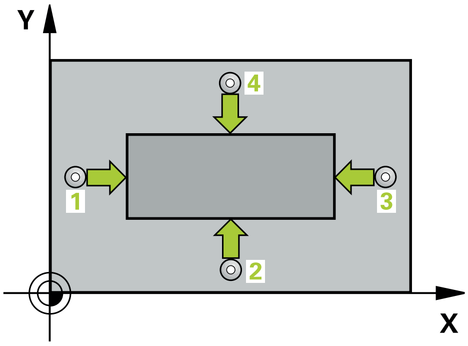
```
11 CYCL DEF 410 PRESET INSIDE RECTAN ~
    Q321=+50            ;CENTER IN 1ST AXIS ~
    Q322=+50            ;CENTER IN 2ND AXIS ~
    Q323=+60            ;FIRST SIDE LENGTH ~
    Q324=+20            ;2ND SIDE LENGTH ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q301=+0             ;MOVE TO CLEARANCE ~
    Q305=+10            ;NUMBER IN TABLE ~
    Q331=+0             ;PRESET ~
    Q332=+0             ;PRESET ~
    Q303=+1             ;MEAS. VALUE TRANSFER ~
    Q381=+1             ;PROBE IN TS AXIS ~
    Q382=+85            ;1ST CO. FOR TS AXIS ~
    Q383=+50            ;2ND CO. FOR TS AXIS ~
    Q384=+0             ;3RD CO. FOR TS AXIS ~
    Q333=+1             ;PRESET
```

29.3.8 Cycle 411 PRESET OUTS. RECTAN

Application

Touch probe cycle **411** finds the center of a rectangular stud and defines this position as the datum. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 The control returns the touch probe to the clearance height.
6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
7 Then the control saves the actual values in the Q parameters listed below.
8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

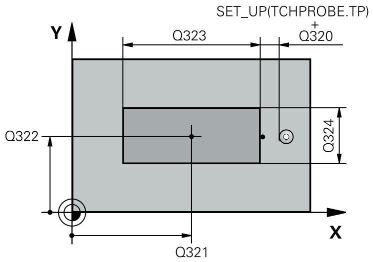
To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic	Parameter



Q321 Center in 1st axis?

Center of the stud in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+9999.9999**

Q322 Center in 2nd axis?

Center of the stud in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q323 First side length?

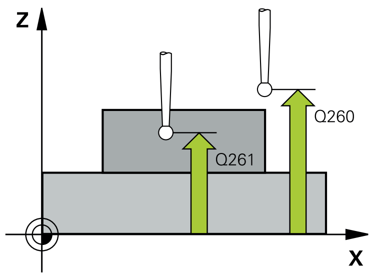
Length of stud parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q324 Second side length?

Length of stud parallel to the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**



Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0**, **1**

Help graphic	Parameter
	Q305 Number in table? Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

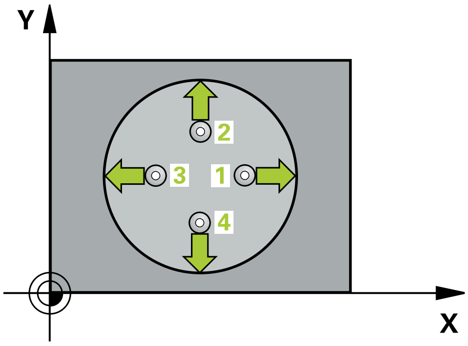
```
11 TCH PROBE 411 PRESET OUTS. RECTAN ~
    Q321=+50                 ;CENTER IN 1ST AXIS ~
    Q322=+50                 ;CENTER IN 2ND AXIS ~
    Q323=+60                 ;FIRST SIDE LENGTH ~
    Q324=+20                 ;2ND SIDE LENGTH ~
    Q261=-5                  ;MEASURING HEIGHT ~
    Q320=+0                  ;SET-UP CLEARANCE ~
    Q260=+20                 ;CLEARANCE HEIGHT ~
    Q301=+0                  ;MOVE TO CLEARANCE ~
    Q305=+0                  ;NUMBER IN TABLE ~
    Q331=+0                  ;PRESET ~
    Q332=+0                  ;PRESET ~
    Q303=+1                  ;MEAS. VALUE TRANSFER ~
    Q381=+1                  ;PROBE IN TS AXIS ~
    Q382=+85                 ;1ST CO. FOR TS AXIS ~
    Q383=+50                 ;2ND CO. FOR TS AXIS ~
    Q384=+0                  ;3RD CO. FOR TS AXIS ~
    Q333=+1                  ;PRESET
```

29.3.9 Cycle 412 PRESET INSIDE CIRCLE

Application

Touch probe cycle **412** finds the center of a circular pocket (hole) and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1. Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456
2. Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3. Then, the touch probe moves in a circular arc either at measuring height or linearly at clearance height to the next touch point **2** and probes again.
4. The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5. The control returns the touch probe to the clearance height.
6. Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
7. Then the control saves the actual values in the Q parameters listed below.
8. If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole). If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.

- Preposition to the pocket center
- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

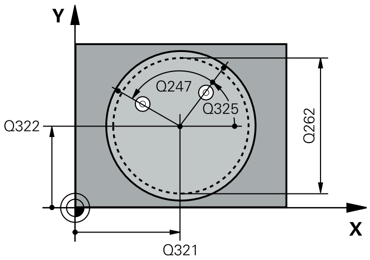
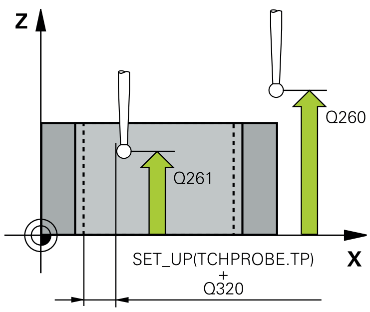
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°

Program the stepping angle to be less than 90°

Cycle parameters

Help graphic	Parameter
	**Q321 Center in 1st axis?** Center of the pocket in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q322 Center in 2nd axis? Center of the pocket in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large. Input: **0...99999.9999**
	Q325 Starting angle? Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect. Input: **-120...+120**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q305 Number in table? Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: **3**: Use three measuring points **4**: Use four measuring points (default setting) Input: **3**, **4**
	Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active. **0**: Move in a straight line between machining operations **1**: Move along a circular arc on the pitch circle diameter between machining operations Input: **0**, **1**

Example

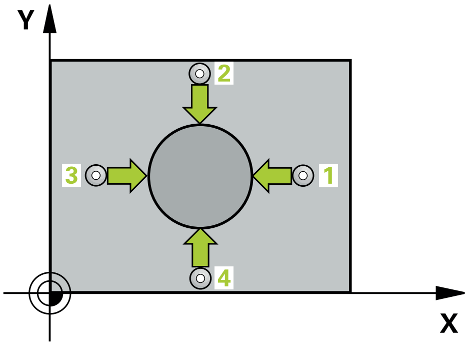
11 TCH PROBE 412 PRESET INSIDE CIRCLE ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+12	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;TYPE OF TRAVERSE

29.3.10 Cycle 413 PRESET OUTS. CIRCLE

Application

Touch probe cycle **413** finds the center of a circular stud and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 The control returns the touch probe to the clearance height.
6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
7 Then the control saves the actual values in the Q parameters listed below.
8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

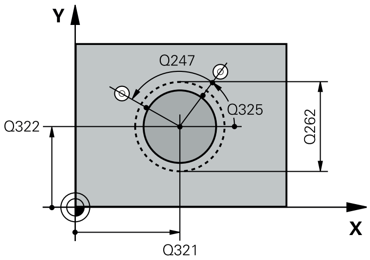
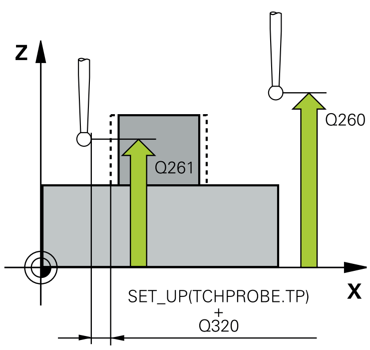
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the nominal diameter of the stud.

- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

- The control will reset an active basic rotation at the beginning of the cycle.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°

Program the stepping angle to be less than 90°

Cycle parameters

Help graphic	Parameter
	**Q321 Center in 1st axis?** Center of the stud in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q322 Center in 2nd axis? Center of the stud in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Approximate diameter of the stud. Enter a value that is more likely to be too large than too small. Input: **0...99999.9999**
	Q325 Starting angle? Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect. Input: **-120...+120**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q305 Number in table? Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: **3**: Use three measuring points **4**: Use four measuring points (default setting) Input: **3**, **4**
	Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active. **0**: Move in a straight line between machining operations **1**: Move along a circular arc on the pitch circle diameter between machining operations Input: **0**, **1**

Example

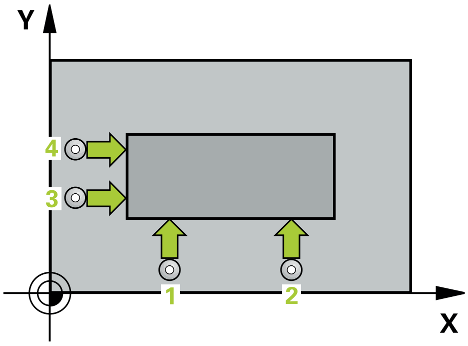
11 TCH PROBE 413 PRESET OUTS. CIRCLE ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+15	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;TYPE OF TRAVERSE

29.3.11 Cycle 414 PRESET OUTS. CORNER

Application

Touch probe cycle **414** finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

Cycle sequence



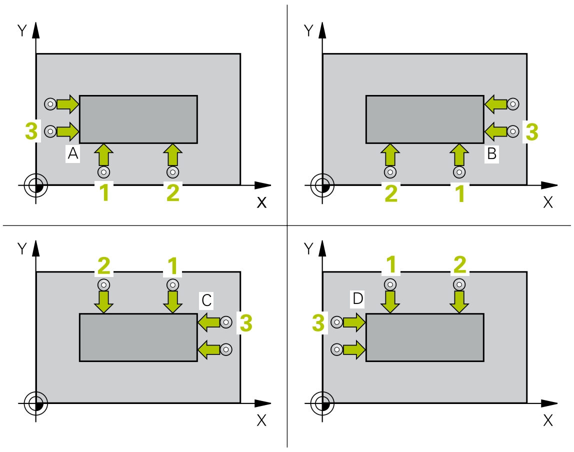
1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) at touch point **1** (see figure). The control offsets the touch probe by the set-up clearance in the direction opposite the respective traverse direction.
   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the 3rd measuring point.
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 The control returns the touch probe to the clearance height.
6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
7 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.
8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

The control always measures the first line in the direction of the minor axis of the working plane.

Q parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Definition of the corner

By defining the positions of the measuring points **1** and **3**, you also determine the corner at which the control sets the preset (see the following figure and table below).



Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater than point **3**

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

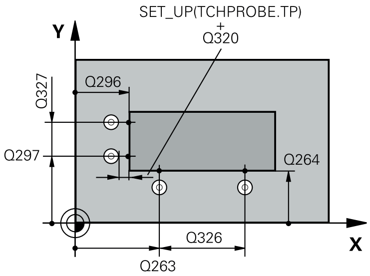
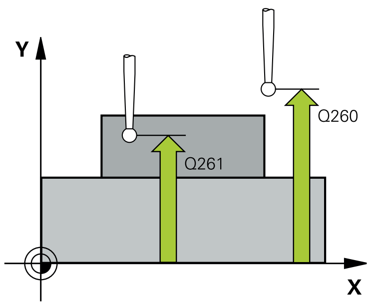
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q326 Spacing in 1st axis? Distance between the first and second measuring points in the main axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q296 3rd measuring point in 1st axis? Coordinate of the third touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q297 3rd measuring point in 2nd axis? Coordinate of the third touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q327 Spacing in 2nd axis? Distance between third and fourth measuring points in the secondary axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q304 Execute basic rotation (0/1)? Define whether the control will compensate workpiece misalignment with a basic rotation: **0**: No basic rotation **1**: Basic rotation Input: **0**, **1**
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table: If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

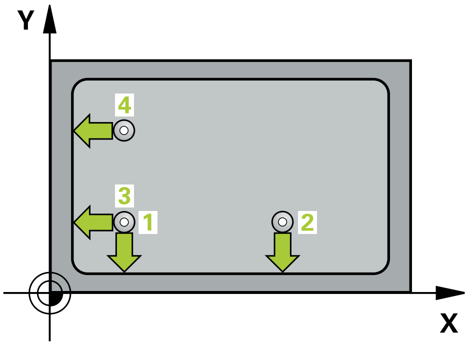
```
11 TCH PROBE 414 PRESET OUTS. CORNER ~
    Q263=+37            ;1ST POINT 1ST AXIS ~
    Q264=+7             ;1ST POINT 2ND AXIS ~
    Q326=+50            ;SPACING IN 1ST AXIS ~
    Q296=+95            ;3RD PNT IN 1ST AXIS ~
    Q297=+25            ;3RD PNT IN 2ND AXIS ~
    Q327=+45            ;SPACING IN 2ND AXIS ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q301=+0             ;MOVE TO CLEARANCE ~
    Q304=+0             ;BASIC ROTATION ~
    Q305=+7             ;NUMBER IN TABLE ~
    Q331=+0             ;PRESET ~
    Q332=+0             ;PRESET ~
    Q303=+1             ;MEAS. VALUE TRANSFER ~
    Q381=+1             ;PROBE IN TS AXIS ~
    Q382=+85            ;1ST CO. FOR TS AXIS ~
    Q383=+50            ;2ND CO. FOR TS AXIS ~
    Q384=+0             ;3RD CO. FOR TS AXIS ~
    Q333=+1             ;PRESET
```

29.3.12 Cycle 415 PRESET INSIDE CORNER

Application

Touch probe cycle **415** finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) at touch point **1** (see figure). The control offsets the touch probe in the main axis and the secondary axis by the set-up clearance **Q320** + **SET_UP** + ball-tip radius (in the direction opposite the respective traverse direction)

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The probing direction is derived from the number by which you identify the corner.
3 The touch probe moves to the next touch point **2**; the control offsets the touch probe in the secondary axis by the set-up clearance **Q320** + **SET_UP** + ball-tip radius and then performs the second probing operation
4 The control positions the touch probe at touch point **3** (same positioning logic as for the first touch point) and performs the probing operation there
5 The touch probe then moves to touch point **4**. The control offsets the touch probe in the main axis by the set-up clearance **Q320** + **SET_UP** + ball-tip radius and then performs the fourth probing operation
6 The control returns the touch probe to the clearance height.
7 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
8 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.
9 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

> The control always measures the first line in the direction of the minor axis of the working plane.

Q parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

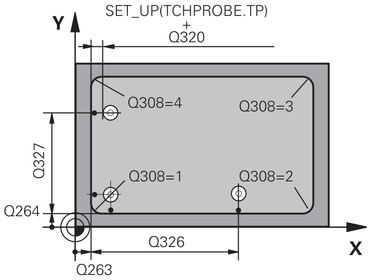
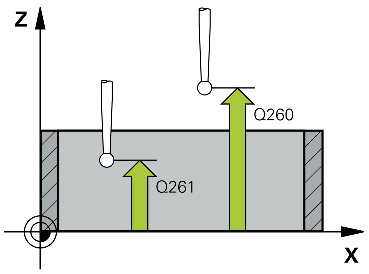
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the corner in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the corner in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q326 Spacing in 1st axis? Distance between the first corner and the second measuring point in the main axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q327 Spacing in 2nd axis? Distance between the corner and the fourth measuring point in the secondary axis of the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q308 Corner? (1/2/3/4) Number identifying the corner at which the control will set the preset. Input: **1**, **2**, **3**, **4**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**

Help graphic	Parameter
	Q304 Execute basic rotation (0/1)? Define whether the control will compensate workpiece misalignment with a basic rotation: **0**: No basic rotation **1**: Basic rotation Input: **0**, **1**
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table: If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

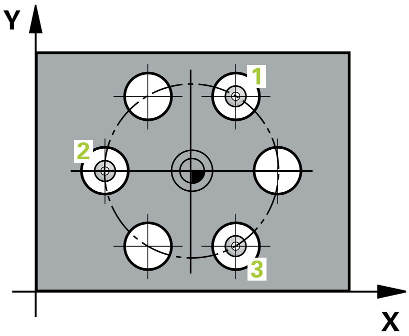
```
11 TCH PROBE 415 PRESET INSIDE CORNER ~
    Q263=+37                ;1ST POINT 1ST AXIS ~
    Q264=+7                 ;1ST POINT 2ND AXIS ~
    Q326=+50                ;SPACING IN 1ST AXIS ~
    Q327=+45                ;SPACING IN 2ND AXIS ~
    Q308=+1                 ;CORNER ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+20                ;CLEARANCE HEIGHT ~
    Q301=+0                 ;MOVE TO CLEARANCE ~
    Q304=+0                 ;BASIC ROTATION ~
    Q305=+7                 ;NUMBER IN TABLE ~
    Q331=+0                 ;PRESET ~
    Q332=+0                 ;PRESET ~
    Q303=+1                 ;MEAS. VALUE TRANSFER ~
    Q381=+1                 ;PROBE IN TS AXIS ~
    Q382=+85                ;1ST CO. FOR TS AXIS ~
    Q383=+50                ;2ND CO. FOR TS AXIS ~
    Q384=+0                 ;3RD CO. FOR TS AXIS ~
    Q333=+1                 ;PRESET
```

29.3.13 Cycle 416 PRESET CIRCLE CENTER

Application

Touch probe cycle **416** finds the center of a bolt hole circle by measuring three holes, and defines the determined center as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed center point of the first hole **1**.

   Further information: "Positioning logic", Page 1456
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
7 The control returns the touch probe to the clearance height.
8 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
9 Then the control saves the actual values in the Q parameters listed below.
10 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

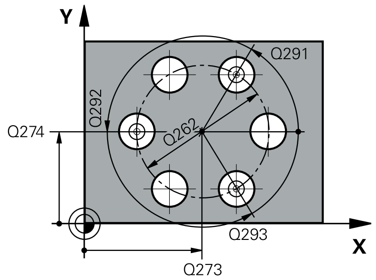
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q273 Center in 1st axis (nom. value)?** Bolt hole circle center (nominal value) in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q274 Center in 2nd axis (nom. value)? Bolt hole circle center (nominal value) in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be. Input: **0...99999.9999**
	Q291 Polar coord. angle of 1st hole? Polar coordinate angle of the first hole center in the working plane. The value has an absolute effect. Input: **-360.000...+360.000**
	Q292 Polar coord. angle of 2nd hole? Polar coordinate angle of the second hole center in the working plane. The value has an absolute effect. Input: **-360.000...+360.000**
	Q293 Polar coord. angle of 3rd hole? Polar coordinate angle of the third hole center in the working plane. The value has an absolute effect. Input: **-360.000...+360.000**
	Q261 Measuring height in probe axis? Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q305 Number in table? Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated bolt-hole center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated bolt-hole circle center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**

Help graphic	Parameter
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Example

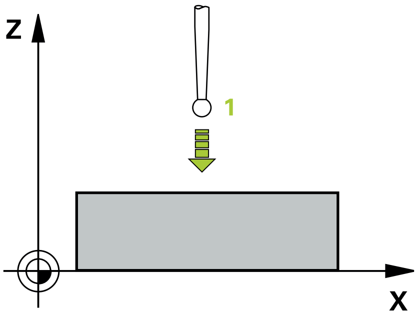
```
11 TCH PROBE 416 PRESET CIRCLE CENTER ~
    Q273=+50                ;CENTER IN 1ST AXIS ~
    Q274=+50                ;CENTER IN 2ND AXIS ~
    Q262=+90                ;NOMINAL DIAMETER ~
    Q291=+34                ;ANGLE OF 1ST HOLE ~
    Q292=+70                ;ANGLE OF 2ND HOLE ~
    Q293=+210               ;ANGLE OF 3RD HOLE ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q260=+20                ;CLEARANCE HEIGHT ~
    Q305=+12                ;NUMBER IN TABLE ~
    Q331=+0                 ;PRESET ~
    Q332=+0                 ;PRESET ~
    Q303=+1                 ;MEAS. VALUE TRANSFER ~
    Q381=+1                 ;PROBE IN TS AXIS ~
    Q382=+85                ;1ST CO. FOR TS AXIS ~
    Q383=+50                ;2ND CO. FOR TS AXIS ~
    Q384=+0                 ;3RD CO. FOR TS AXIS ~
    Q333=+1                 ;PRESET ~
    Q320=+0                 ;SET-UP CLEARANCE
```

29.3.14 Cycle 417 PRESET IN TS AXIS

Application

Touch probe cycle **417** measures any coordinate in the touch probe axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction of the positive touch probe axis.

   Further information: "Positioning logic", Page 1456
2 Then the touch probe moves in its own axis to the coordinate entered as touch point **1** and measures the actual position with a simple probing movement
3 The control returns the touch probe to the clearance height.
4 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
5 Then the control saves the actual values in the Q parameters listed below.

Q parameter number	Meaning
Q160	Actual value of measured point

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

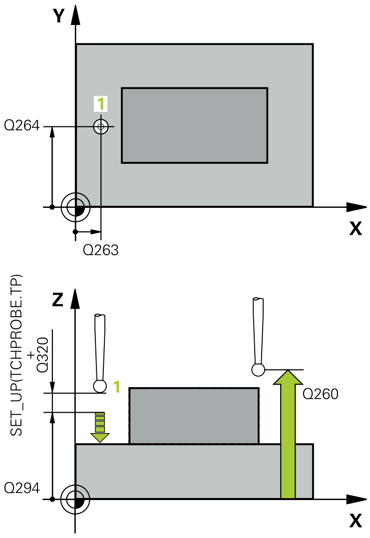
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control sets the preset in this axis.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Example

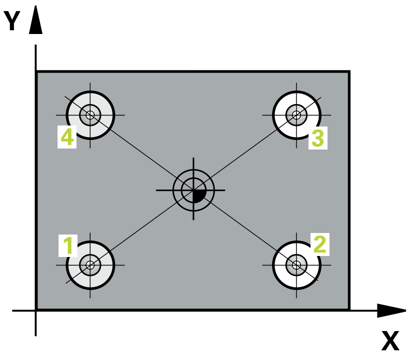
```
11 TCH PROBE 417 PRESET IN TS AXIS ~
    Q263=+25                 ;1ST POINT 1ST AXIS ~
    Q264=+25                 ;1ST POINT 2ND AXIS ~
    Q294=+25                 ;1ST POINT 3RD AXIS ~
    Q320=+0                  ;SET-UP CLEARANCE ~
    Q260=+50                 ;CLEARANCE HEIGHT ~
    Q305=+0                  ;NUMBER IN TABLE ~
    Q333=+0                  ;PRESET ~
    Q303=+1                  ;MEAS. VALUE TRANSFER
```

29.3.15 Cycle 418 PRESET FROM 4 HOLES

Application

Touch probe cycle **418** calculates the intersection of the lines connecting two opposite hole center points and sets the preset at the point of intersection. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the center point of the first hole **1**.
   Further information: "Positioning logic", Page 1456
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The control repeats this step for holes **3** and **4**.
6 The control returns the touch probe to the clearance height.
7 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
8 The control calculates the preset as the intersection of the lines connecting the centers of holes **1**/**3** and **2**/**4** and saves the actual values in the Q parameters listed below.
9 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

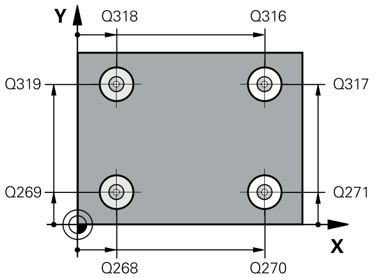
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

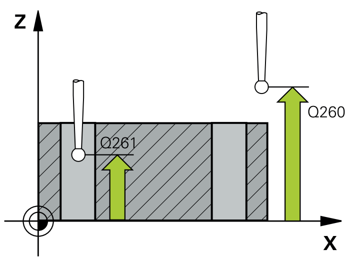
Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic





Parameter

Q268 1st hole: center in 1st axis?
Center of the first hole in the main axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+9999.9999**

Q269 1st hole: center in 2nd axis?
Center of the first hole in the secondary axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q270 2nd hole: center in 1st axis?
Center of the second hole in the main axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q271 2nd hole: center in 2nd axis?
Center of the second hole in the secondary axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q316 3rd hole: Center in 1st axis?
Center of the third hole in the main axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q317 3rd hole: Center in 2nd axis?
Center of the third hole in the secondary axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q318 4th hole: Center in 1st axis?
Center of the fourth hole in the main axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q319 4th hole: Center in 2nd axis?
Center of the fourth hole in the secondary axis of the working plane. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.
Input: **-99999.9999...+99999.9999**

Q260 Clearance height?
Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.
Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q305 Number in table? Indicate the number of the row in the preset table or datum table in which the control saves the coordinates of the point of intersection of the connecting lines. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**

Help graphic	Parameter
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

```
11 TCH PROBE 418 PRESET FROM 4 HOLES ~
    Q268=+20            ;1ST CENTER 1ST AXIS ~
    Q269=+25            ;1ST CENTER 2ND AXIS ~
    Q270=+150           ;2ND CENTER 1ST AXIS ~
    Q271=+25            ;2ND CENTER 2ND AXIS ~
    Q316=+150           ;3RD CENTER 1ST AXIS ~
    Q317=+85            ;3RD CENTER 2ND AXIS ~
    Q318=+22            ;4TH CENTER 1ST AXIS ~
    Q319=+80            ;4TH CENTER 2ND AXIS ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q260=+10            ;CLEARANCE HEIGHT ~
    Q305=+12            ;NUMBER IN TABLE ~
    Q331=+0             ;PRESET ~
    Q332=+0             ;PRESET ~
    Q303=+1             ;MEAS. VALUE TRANSFER ~
    Q381=+1             ;PROBE IN TS AXIS ~
    Q382=+85            ;1ST CO. FOR TS AXIS ~
    Q383=+50            ;2ND CO. FOR TS AXIS ~
    Q384=+0             ;3RD CO. FOR TS AXIS ~
    Q333=+0             ;PRESET
```

29.3.16 Cycle 419 PRESET IN ONE AXIS

Application

Touch probe cycle **419** measures any coordinate in the a selectable axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.

Cycle sequence

1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the programmed probing direction.

   Further information: "Positioning logic", Page 1456

2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.
3 The control returns the touch probe to the clearance height.
4 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

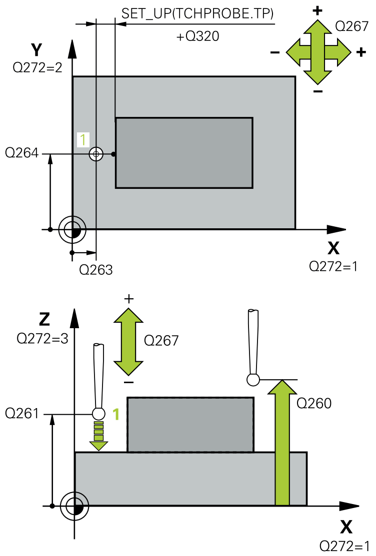
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you want to save the preset in several axes in the preset table, you can use Cycle **419** several times in a row. However, you also have to reactivate the preset number after every run of Cycle **419**. If you work with preset 0 as active preset, this process is not required.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

1: Main axis = measuring axis

2: Secondary axis = measuring axis

3: Touch probe axis = measuring axis

Axis assignment

Active touch probe axis: Q272 = 3	Correspond-ing main axis: Q272 = 1	Corresponding secondary axis: Q272 = 2
Z	X	Y
Y	Z	X
X	Y	Z

Input: **1**, **2**, **3**

Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

-1: Negative traverse direction

+1: Positive traverse direction

Input: **-1**, **+1**

Help graphic	Parameter
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539
	Q333 New preset? Coordinate at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **-1**: Do not use. Is entered by the control when old NC programs are read in, (see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 1538) **0**: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **-1**, **0**, **+1**

Example

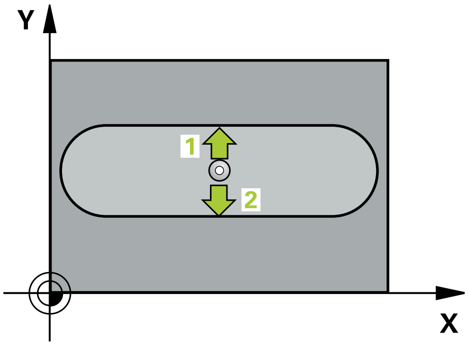
```
11 TCH PROBE 419 PRESET IN ONE AXIS ~
    Q263=+25                ;1ST POINT 1ST AXIS ~
    Q264=+25                ;1ST POINT 2ND AXIS ~
    Q261=+25                ;MEASURING HEIGHT ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+50                ;CLEARANCE HEIGHT ~
    Q272=+1                 ;MEASURING AXIS ~
    Q267=+1                 ;TRAVERSE DIRECTION ~
    Q305=+0                 ;NUMBER IN TABLE ~
    Q333=+0                 ;PRESET ~
    Q303=+1                 ;MEAS. VALUE TRANSFER
```

29.3.17 Cycle 408 SLOT CENTER PRESET

Application

Touch probe cycle **408** finds the center of a slot and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height.
5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
6 Then the control saves the actual values in the Q parameters listed below.
7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q166	Actual value of measured slot width
Q157	Actual value of the centerline

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

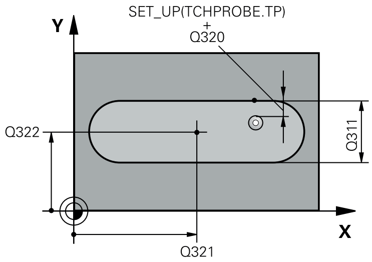
To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width. If the slot width and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the slot. In this case, the touch probe does not return to the clearance height between the two measuring points.

- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic	Parameter



Q321 Center in 1st axis?

Center of the slot in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q322 Center in 2nd axis?

Center of the slot in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q311 Width of slot?

Width of the slot, regardless of its position in the working plane. This value has an incremental effect.

Input: **0...99999.9999**

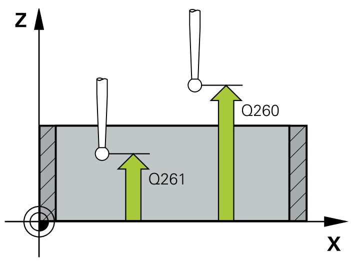
Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

1: Main axis = measuring axis

2: Secondary axis = measuring axis

Input: **1**, **2**



Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0**, **1**

Help graphic	Parameter
	Q305 Number in table? Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q405 New preset? Coordinate in the measuring axis at which the control will set the calculated slot center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+9999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **0**: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **0**, **1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

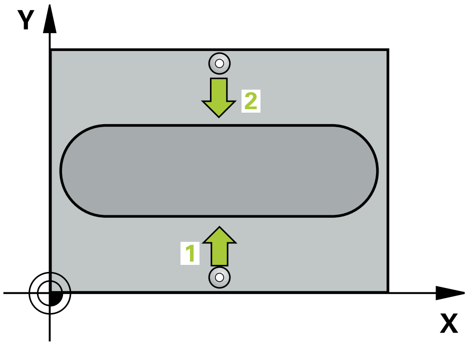
```
11 TCH PROBE 408 SLOT CENTER PRESET ~
    Q321=+50        ;CENTER IN 1ST AXIS ~
    Q322=+50        ;CENTER IN 2ND AXIS ~
    Q311=+25        ;SLOT WIDTH ~
    Q272=+1         ;MEASURING AXIS ~
    Q261=-5         ;MEASURING HEIGHT ~
    Q320=+0         ;SET-UP CLEARANCE ~
    Q260=+20        ;CLEARANCE HEIGHT ~
    Q301=+0         ;MOVE TO CLEARANCE ~
    Q305=+10        ;NUMBER IN TABLE ~
    Q405=+0         ;PRESET ~
    Q303=+1         ;MEAS. VALUE TRANSFER ~
    Q381=+1         ;PROBE IN TS AXIS ~
    Q382=+85        ;1ST CO. FOR TS AXIS ~
    Q383=+50        ;2ND CO. FOR TS AXIS ~
    Q384=+0         ;3RD CO. FOR TS AXIS ~
    Q333=+1         ;PRESET
```

29.3.18 Cycle 409 RIDGE CENTER PRESET

Application

Touch probe cycle **409** finds the center of a ridge and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves at clearance height to the next touch point **2** and probes it.
4 The control returns the touch probe to the clearance height.
5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, (see "Fundamentals of touch probe cycles 4xx for preset setting", Page 1538)
6 Then the control saves the actual values in the Q parameters listed below.
7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Q parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

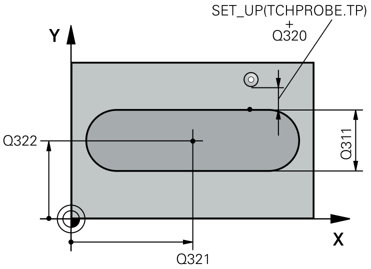
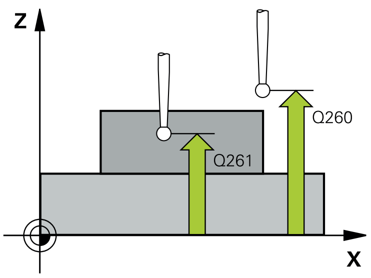
Notes

NOTICE
Danger of collision! When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active. ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**. ▶ Reset any coordinate transformations beforehand.

NOTICE
Danger of collision! To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width. ▶ Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Cycle parameters

Help graphic	Parameter
 	**Q321 Center in 1st axis?** Center of the ridge in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q322 Center in 2nd axis? Center of the ridge in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q311 Ridge width? Width of the ridge, regardless of its position in the working plane. This value has an incremental effect. Input: **0...99999.9999**
	Q272 Measuring axis (1=1st / 2=2nd)? Axis in the working plane in which the measurement will be performed: **1**: Main axis = measuring axis **2**: Secondary axis = measuring axis Input: **1**, **2**
 	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

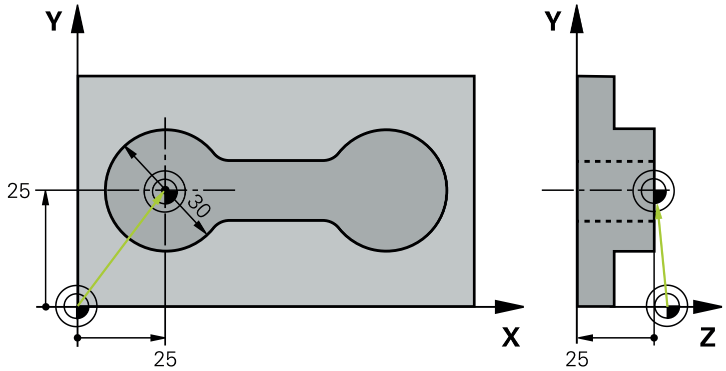
Help graphic	Parameter
	Q305 Number in table? Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table. If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation. If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated. **Further information:** "Saving the calculated preset", Page 1539 Input: **0...99999**
	Q405 New preset? Coordinate in the measuring axis at which the control will set the calculated ridge center. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: **0**: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system. **1**: Write the calculated preset to the preset table. Input: **0**, **1**
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: **0**: Do not set the preset in the touch probe axis **1**: Set the preset in the touch probe axis Input: **0**, **1**
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Example

```
11 TCH PROBE 409 RIDGE CENTER PRESET ~
    Q321=+50            ;CENTER IN 1ST AXIS ~
    Q322=+50            ;CENTER IN 2ND AXIS ~
    Q311=+25            ;RIDGE WIDTH ~
    Q272=+1             ;MEASURING AXIS ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q305=+10            ;NUMBER IN TABLE ~
    Q405=+0             ;PRESET ~
    Q303=+1             ;MEAS. VALUE TRANSFER ~
    Q381=+1             ;PROBE IN TS AXIS ~
    Q382=+85            ;1ST CO. FOR TS AXIS ~
    Q383=+50            ;2ND CO. FOR TS AXIS ~
    Q384=+0             ;3RD CO. FOR TS AXIS ~
    Q333=+1             ;PRESET
```

29.3.19 Example: Presetting at center of a circular segment and on top surface of workpiece

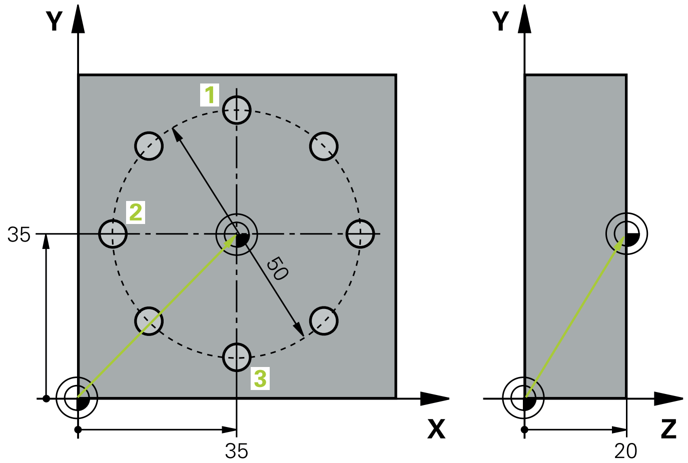


- **Q325** = Polar coordinate angle for touch point 1
- **Q247** = Stepping angle for calculating the touch points 2 to 4
- **Q305** = Write to row number 5 of the preset table
- **Q303** = Write the calculated preset to the preset table
- **Q381** = Also set the preset in the touch probe axis
- **Q365** = Move on circular path between measuring points

```
0  BEGIN PGM 413 MM
1  TOOL CALL "TOUCH_PROBE" Z
2  TCH PROBE 413 PRESET OUTS. CIRCLE ~
    Q321=+25            ;CENTER IN 1ST AXIS ~
    Q322=+25            ;CENTER IN 2ND AXIS ~
    Q262=+30            ;NOMINAL DIAMETER ~
    Q325=+90            ;STARTING ANGLE ~
    Q247=+45            ;STEPPING ANGLE ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q320=+2             ;SET-UP CLEARANCE ~
    Q260=+50            ;CLEARANCE HEIGHT ~
    Q301=+0             ;MOVE TO CLEARANCE ~
    Q305=+5             ;NUMBER IN TABLE ~
    Q331=+0             ;PRESET ~
    Q332=+10            ;PRESET ~
    Q303=+1             ;MEAS. VALUE TRANSFER ~
    Q381=+1             ;PROBE IN TS AXIS ~
    Q382=+25            ;1ST CO. FOR TS AXIS ~
    Q383=+25            ;2ND CO. FOR TS AXIS ~
    Q384=+0             ;3RD CO. FOR TS AXIS ~
    Q333=+0             ;PRESET ~
    Q423=+4             ;NO. OF PROBE POINTS ~
    Q365=+0             ;TYPE OF TRAVERSE
3  END PGM 413 MM
```

29.3.20 Example: Presetting on top surface of workpiece and at center of a bolt hole circle

The control will write the measured bolt-hole circle center to the preset table so that it may be used at a later time.



- **Q291** = Polar coordinate angle for first hole center **1**
- **Q292** = Polar coordinate angle for second hole center **2**
- **Q293** = Polar coordinate angle for third hole center **3**
- **Q305** = Write center of bolt hole circle (X and Y) to row 1
- **Q303** = In the preset table **PRESET.PR**, save the calculated preset referenced to the machine-based coordinate system (REF system)

```
0  BEGIN PGM 416 MM
1  TOOL CALL "TOUCH_PROBE" Z
2  TCH PROBE 416 PRESET CIRCLE CENTER ~
    Q273=+35              ;CENTER IN 1ST AXIS ~
    Q274=+35              ;CENTER IN 2ND AXIS ~
    Q262=+50              ;NOMINAL DIAMETER ~
    Q291=+90              ;ANGLE OF 1ST HOLE ~
    Q292=+180             ;ANGLE OF 2ND HOLE ~
    Q293=+270             ;ANGLE OF 3RD HOLE ~
    Q261=+15              ;MEASURING HEIGHT ~
    Q260=+10              ;CLEARANCE HEIGHT ~
    Q305=+1               ;NUMBER IN TABLE ~
    Q331=+0               ;PRESET ~
    Q332=+0               ;PRESET ~
    Q303=+1               ;MEAS. VALUE TRANSFER ~
    Q381=+1               ;PROBE IN TS AXIS ~
    Q382=+7.5             ;1ST CO. FOR TS AXIS ~
    Q383=+7.5             ;2ND CO. FOR TS AXIS ~
    Q384=+20              ;3RD CO. FOR TS AXIS ~
    Q333=+0               ;PRESET ~
    Q320=+0               ;SET-UP CLEARANCE.
3  CYCL DEF 247 PRESETTING ~
    Q339=+1               ;PRESET NUMBER
4  END PGM 416 MM
```

29.4 Touch Probe Cycles: Automatic Workpiece Inspection

29.4.1 Fundamentals

Overview

The control must be specifically prepared by the machine manufacturer for the use of a touch probe.

HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

The control offers cycles for measuring workpieces automatically:

Cycle		Call	Further information
0	**REF. PLANE** ■ Measuring a coordinate in a selectable axis	**DEF-**active	Page 1611
1	**POLAR PRESET** ■ Measuring a point ■ Probing direction via angle	**DEF-**active	Page 1612
420	**MEASURE ANGLE** ■ Measuring an angle in the working plane	**DEF-**active	Page 1614
421	**MEASURE HOLE** ■ Measuring the position of a hole ■ Measuring the diameter of a hole ■ Nominal-to-actual value comparison, if applicable	**DEF-**active	Page 1617
422	**MEAS. CIRCLE OUTSIDE** ■ Measuring the position of a circular stud ■ Measuring the diameter of a circular stud ■ Nominal-to-actual value comparison, if applicable	**DEF-**active	Page 1623
423	**MEAS. RECTAN. INSIDE** ■ Measuring the position of a rectangular pocket ■ Measuring the length and width of a rectangular pocket ■ Nominal-to-actual value comparison, if applicable	**DEF-**active	Page 1629
424	**MEAS. RECTAN. OUTS.** ■ Measuring the position of a rectangular stud ■ Measuring the length and width of a rectangular stud ■ Nominal-to-actual value comparison, if applicable	**DEF-**active	Page 1633
425	**MEASURE INSIDE WIDTH** ■ Measuring the position of a slot ■ Measuring the width of a slot ■ Nominal-to-actual value comparison, if applicable	**DEF-**active	Page 1638
426	**MEASURE RIDGE WIDTH** ■ Measuring the position of a ridge ■ Measuring the width of a ridge ■ Nominal-to-actual value comparison, if applicable	**DEF-**active	Page 1642

Cycle		Call	Further information
427	**MEASURE COORDINATE** ■ Measuring any coordinate in a selectable axis ■ Nominal-to-actual value comparison, if applicable	**DEF**-active	Page 1645
430	**MEAS. BOLT HOLE CIRC** ■ Measuring the center point of a bolt hole circle ■ Measuring the diameter of a bolt hole circle ■ Nominal-to-actual value comparison, if applicable	**DEF**-active	Page 1650
431	**MEASURE PLANE** ■ Finding the angle of a plane by measuring three points	**DEF**-active	Page 1655

Recording the results of measurement

For all cycles in which you automatically measure workpieces (with the exception of Cycles **0** and **1**), you can have the control record the measurement results in a log. In the respective probing cycle you can define if the control is to

- Save the measuring log to a file
- Interrupt program run and display the measuring log on the screen
- Create no measuring log

If you want to save the measuring log to a file, the control by default saves the data as an ASCII file. The control will save the file in the directory that also contains the associated NC program.

The unit of measurement of the main program can be seen in the header of the log file.

> Use the HEIDENHAIN data transfer software TNCremo if you wish to output the measuring log over the data interface.

Example: Measuring log for touch probe cycle **421**:

Measuring log for Probing Cycle 421 Hole Measuring

Date: 30-06-2005
Time: 6:55:04
Measuring program: TNC:\GEH35712\CHECK1.H
Type of dimension (0 = MM / 1 = INCH): 0

Nominal values:	
Center in reference axis:	50.0000
Center in minor axis:	65.0000
Diameter:	12.0000
Given limit values:	
Maximum limit for center in reference axis:	50.1000
Minimum limit for center in reference axis:	49.9000
Maximum limit for center in minor axis:	65.1000
Minimum limit for center in minor axis:	64.9000
Maximum dimension for hole:	12.0450
Minimum dimension for hole:	12.0000
Actual values:	
Center in reference axis:	50.0810
Center in minor axis:	64.9530
Diameter:	12.0259
Deviations:	
Center in reference axis:	0.0810
Center in minor axis:	-0.0470
Diameter:	0.0259
Further measuring results: Measuring height:	-5.0000

End of measuring log

Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. Deviations from the nominal values are saved in parameters **Q161** to **Q166**. Note the table of result parameters listed with every cycle description.

During cycle definition, the control also shows the result parameters for the respective cycle in a help graphic . The highlighted result parameter belongs to that input parameter.

Classification of results

For some cycles you can inquire the status of measuring results through the globally effective Q parameters **Q180** to **Q182**.

Parameter value	Measuring status
Q180 = 1	Measurement results are within tolerance
Q181 = 1	Rework is required
Q182 = 1	Scrap

The control sets the rework or scrap marker as soon as one of the measuring values is out of tolerance. To determine which of the measuring results is out of tolerance, check the measuring log, or compare the respective measuring results (**Q150** to **Q160**) with their limit values.

In Cycle **427** the control assumes by default that you are measuring an outside dimension (stud). However, you can correct the status of the measurement by entering the correct maximum and minimum dimension together with the probing direction.

> The control also sets the status markers if you have not defined any tolerance values or maximum/minimum dimensions.

Tolerance monitoring

With most cycles for workpiece inspection, you can have the control perform tolerance monitoring. This requires that you define the necessary limit values during cycle definition. If you do not wish to monitor for tolerances, simply leave the default value 0 for this parameter set this parameter unchanged.

Tool monitoring

With some cycles for workpiece inspection, you can have the control perform tool monitoring. The control then monitors whether

- the tool radius should be compensated due to the deviations from the nominal value (values in **Q16x**)
- the deviations from the nominal value (values in **Q16x**) are greater than the tool breakage tolerance.

Tool compensation

Requirements:

- Active tool table
- Tool monitoring must be switched on in the cycle: Set **Q330** unequal to 0 or enter a tool name. Select the tool name input via **Name** in the action bar.

> - HEIDENHAIN recommends using this function only if the tool to be compensated for is the one that was used to machine the contour as well as if any necessary reworking will also be done with this tool.
> - If you perform several compensation measurements, the control adds the respective measured deviation to the value stored in the tool table.

Milling cutter

If you reference a milling cutter in parameter **Q330**, the appropriate values will be compensated as follows:

The control always compensates the tool radius in the **DR** column of the tool table, even if the measured deviation lies within the given tolerance.

You can inquire whether re-working is necessary via parameter **Q181** in the NC program (**Q181**=1: rework required).

Turning tool

Only applies to Cycles **421**, **422**, **427**.

If you reference a turning tool in parameter **Q330**, the appropriate values in row DZL and DXL, respectively, will be compensated. The control also monitors the breakage tolerance, which is defined in column LBREAK.

You can inquire whether re-working is necessary via parameter **Q181** in the NC program (**Q181**=1: rework required).

Compensating an indexed tool

If you want to automatically compensate the values for an indexed tool with a tool name, program the following:

- **QS0** = "TOOL NAME"
- **FN18: SYSREAD Q0 = ID990 NR10 IDX0**; specify the number of the **QS** parameter in **IDX**
- **Q0**= **Q0** +0.2; add the index of the basic tool number
- In the cycle: **Q330** = **Q0**; use the indexed tool number

Tool breakage monitoring

Requirements:

- Active tool table
- Tool monitoring must be switched on in the cycle (set **Q330** unequal to 0)
- RBREAK must be greater than 0 (in the entered tool number in the table)

  Further information: "Tool data", Page 249

The control will output an error message and stop the program run if the measured deviation is greater than the breakage tolerance of the tool. At the same time, the tool will be deactivated in the tool table (column TL = L).

Reference system for measurement results

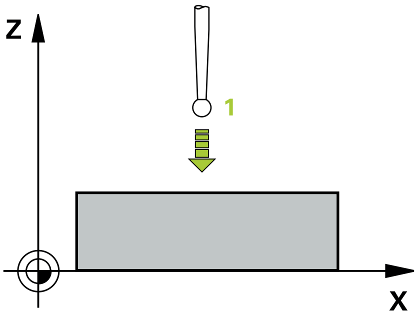
The control transfers all measurement results, which reference the active coordinate system, or as the case may be, the shifted or/and rotated/tilted coordinate system, to the result parameters and the log file.

29.4.2 Cycle 0 REF. PLANE

Application

The touch probe cycle measures any position on the workpiece in a selectable axis direction.

Cycle sequence



1. In a 3-D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
2. Next, the touch probe performs probing at the probing feed rate (**F** column). The probing direction must be defined in the cycle.
3. After the control has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. In addition, the control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**. For the values in these parameters the control does not account for the stylus length and radius.

Notes

NOTICE

Danger of collision!

The control moves the touch probe in a 3-D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- Pre-position to a position where there is no danger of collision when the programmed pre-positioning point is approached

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

Cycle parameters

Help graphic	Parameter
	Parameter number for result? Enter the number of the Q parameter to which you want to assign the coordinate.. Input: **0...1999**
	Probing axis/probing direction? Select the probing axis with the axis key or the alphabetic keyboard, entering the algebraic sign for the probing direction. Input: **-, +**
	Position value? Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input: **-999999999...+999999999**

Example

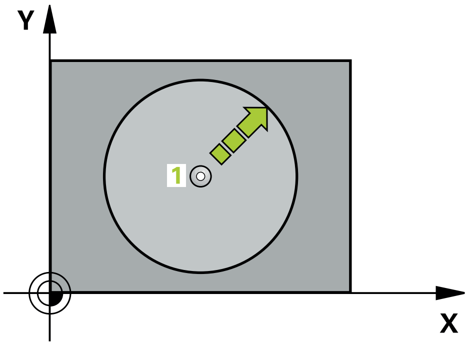
```
11 TCH PROBE 0.0 REF. PLANE Q9 Z+
12 TCH PROBE 0.1  X+99  Y+22  Z+2
```

29.4.3 Cycle 1 POLAR PRESET

Application

Touch probe cycle **1** measures any position on the workpiece in any probing direction.

Cycle sequence



1 In a 3-D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
2 Next, the touch probe performs probing at the probing feed rate (**F** column). During probing, the control moves the touch probe simultaneously in two axes (depending on the probing angle). Use polar angles to define the probing direction in the cycle.
3 After the control has saved the position, the touch probe returns to the starting point. The control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**

Notes

NOTICE

Danger of collision!

The control moves the touch probe in a 3-D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- Pre-position to a position where there is no danger of collision when the programmed pre-positioning point is approached

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The probing axis defined in the cycle specifies the probing plane:
  Probing axis X: X/Y plane
  Probing axis Y: Y/Z plane
  Probing axis Z: Z/X plane

Cycle parameters

Help graphic	Parameter
	Probing axis? Enter the probing axis with the axis key or the alphabetic keyboard. Confirm with the **ENT** key. Input: **X**, **Y**, or **Z**
	Probing angle? Angle measured from the probing axis in which the touch probe will move. Input: **-180...+180**
	Position value? Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input: **-999999999...+999999999**

Example

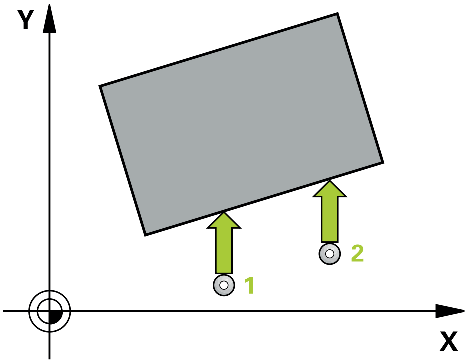
```
11 TCH PROBE 1.0 POLAR PRESET
12 TCH PROBE 1.1 X WINKEL:+30
13 TCH PROBE 1.2  X+0  Y+10  Z+3
```

29.4.4 Cycle 420 MEASURE ANGLE

Application

Touch probe cycle **420** measures the angle that any straight line on the workpiece forms with the main axis of the working plane.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account for probe movements in any probing direction. When the probe movement starts, the center of the ball tip will be offset by this sum in the direction opposite the probing direction.

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 The touch probe then moves to the next touch point **2** and probes again.
4 The control returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

Q parameter number	Meaning
Q150	The measured angle is referenced to the reference axis of the working plane.

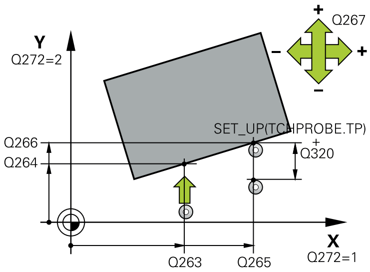
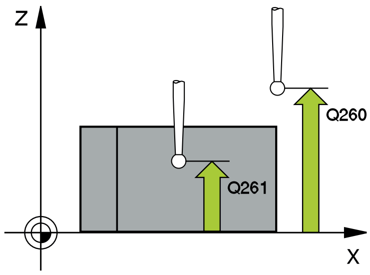
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:
  - If you want to measure the angle in the direction of the A axis, set **Q263** equal to **Q265** and **Q264** unequal to **Q266**.
  - If you want to measure the angle in the direction of the B axis, set **Q263** not equal to **Q265** and **Q264** equal to **Q266**.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q265 2nd measuring point in 1st axis? Coordinate of the second touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q266 2nd measuring point in 2nd axis? Coordinate of the second touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q272 Meas. axis (1/2/3, 1=ref. axis)? Axis in which the measurement will be made: **1**: Main axis = measuring axis **2**: Secondary axis = measuring axis **3**: Touch probe axis = measuring axis Input: **1**, **2**, **3**
	Q267 Trav. direction 1 (+1=+ / -1=-)? Direction in which the touch probe will approach the workpiece: **-1**: Negative traverse direction **+1**: Positive traverse direction Input: **-1**, **+1**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between measuring point and ball tip. The touch probe movement will start with an offset of the sum of **Q320**, **SET_UP**, and the ball-tip radius, even when probing in the tool axis direction. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: Define whether the control will create a measuring log: **1**: Create a measuring log: The control will save the **log file named TCHPR420.TXT** in the folder that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen (you can later resume the NC program run with **NC Start**) Input: **0**, **1**, **2**

Example

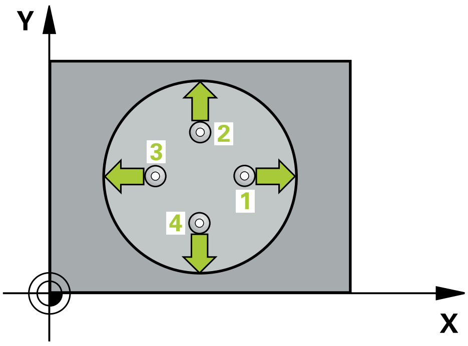
```
11 TCH PROBE 420 MEASURE ANGLE ~
    Q263=+10              ;1ST POINT 1ST AXIS ~
    Q264=+10              ;1ST POINT 2ND AXIS ~
    Q265=+15              ;2ND PNT IN 1ST AXIS ~
    Q266=+95              ;2ND POINT 2ND AXIS ~
    Q272=+1               ;MEASURING AXIS ~
    Q267=-1               ;TRAVERSE DIRECTION ~
    Q261=-5               ;MEASURING HEIGHT ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q260=+10              ;CLEARANCE HEIGHT ~
    Q301=+1               ;MOVE TO CLEARANCE ~
    Q281=+1               ;MEASURING LOG
```

29.4.5 Cycle 421 MEASURE HOLE

Application

Touch probe cycle **421** measures the center point and diameter of a hole (or circular pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the SET_UP column of the touch probe table.
   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

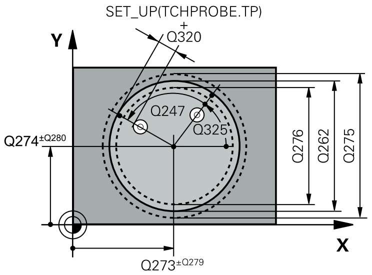
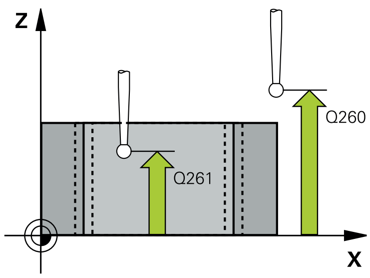
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter Q330 references a turning tool, the following applies:
  - Parameters **Q498** and **Q531** have to be defined
  - The information in parameters **Q498**, **Q531**, for example from Cycle **800**, has to match this information
  - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
  - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

Help graphic	Parameter
	**Q273 Center in 1st axis (nom. value)?** Center of the hole in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q274 Center in 2nd axis (nom. value)? Center of the hole in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Enter the diameter of the hole. Input: **0...99999.9999**
	Q325 Starting angle? Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: **-360.000...+360.000**
	Q247 Intermediate stepping angle? Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect. Input: **-120...+120**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**

Help graphic	Parameter
	Q275 Maximum limit of size for hole? Maximum permissible diameter for the hole (circular pocket) Input: **0...99999.9999**
	Q276 Minimum limit of size? Minimum permissible diameter for the hole (circular pocket) Input: **0...99999.9999**
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: **0...99999.9999**
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR421.TXT** by default in the directory that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control will monitor the tool Page 1609: **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters

Help graphic	Parameter
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: **3**: Use three measuring points **4**: Use four measuring points (default setting) Input: **3**, **4**
	Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active. **0**: Move in a straight line between machining operations **1**: Move along a circular arc on the pitch circle diameter between machining operations Input: **0**, **1**
	Q498 Reverse tool (0=no/1=yes)? Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following: **1**: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 1 **0**: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example , Cycle **800** and parameter **Reverse the tool Q498** = 0 Input: **0**, **1**
	Q531 Angle of incidence? Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**). Input: **-180...+180**

Example

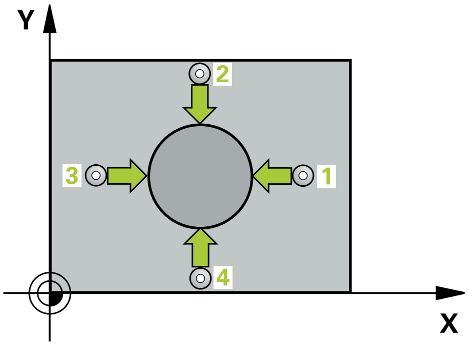
```
11 TCH PROBE 421 MEASURE HOLE ~
    Q273=+50                ;CENTER IN 1ST AXIS ~
    Q274=+50                ;CENTER IN 2ND AXIS ~
    Q262=+75                ;NOMINAL DIAMETER ~
    Q325=+0                 ;STARTING ANGLE ~
    Q247=+60                ;STEPPING ANGLE ~
    Q261=-5                 ;MEASURING HEIGHT ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q260=+20                ;CLEARANCE HEIGHT ~
    Q301=+1                 ;MOVE TO CLEARANCE ~
    Q275=+75.12             ;MAXIMUM LIMIT ~
    Q276=+74.95             ;MINIMUM LIMIT ~
    Q279=+0.1               ;TOLERANCE 1ST CENTER ~
    Q280=+0.1               ;TOLERANCE 2ND CENTER ~
    Q281=+1                 ;MEASURING LOG ~
    Q309=+0                 ;PGM STOP TOLERANCE ~
    Q330=+0                 ;TOOL ~
    Q423=+4                 ;NO. OF PROBE POINTS ~
    Q365=+1                 ;TYPE OF TRAVERSE ~
    Q498=+0                 ;REVERSE TOOL ~
    Q531=+0                 ;ANGLE OF INCIDENCE
```

29.4.6 Cycle 422 MEAS. CIRCLE OUTSIDE

Application

Touch probe cycle **422** measures the center point and diameter of a circular stud. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Notes

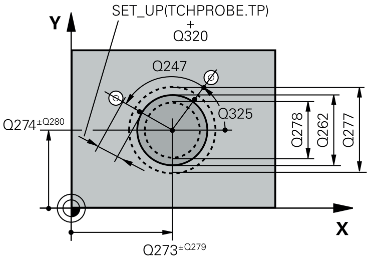
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter Q330 references a turning tool, the following applies:
  - Parameters **Q498** and **Q531** have to be defined
  - The information in parameters **Q498**, **Q531**, for example from Cycle **800**, has to match this information
  - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
  - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

Help graphic | **Parameter**



Q273 Center in 1st axis (nom. value)?

Center of the stud in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Center of the stud in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Enter the diameter of the stud.

Input: **0...99999.9999**

Q325 Starting angle?

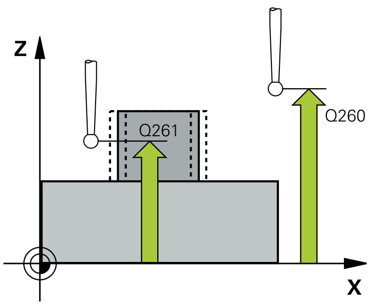
Angle between the main axis of the working plane and the first touch point. The value has an absolute effect.

Input: **-360.000...+360.000**

Q247 Intermediate stepping angle?

Angle between two measuring points. The algebraic sign of the stepping angle determines the machining direction (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. This value has an incremental effect.

Input: **-120...+120**



Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0**, **1**

Help graphic	Parameter
	Q277 Maximum limit of size for stud? Maximum permissible diameter for the stud. Input: **0...99999.9999**
	Q278 Minimum limit of size for stud? Minimum permissible diameter for the stud. Input: **0...99999.9999**
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: **0...99999.9999**
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR422.TXT** in the folder that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control will monitor the tool Page 1609. **0**: Monitoring not active **> 0**: Tool number in tool table TOOL.T Input: **0...99999.9** or max. **255** characters
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: **3**: Use three measuring points **4**: Use four measuring points (default setting) Input: **3**, **4**

Help graphic	Parameter
	Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (**Q301** = 1) is active. **0**: Move in a straight line between machining operations **1**: Move along a circular arc on the pitch circle diameter between machining operations Input: **0**, **1**
	Q498 Reverse tool (0=no/1=yes)? Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following: **1**: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 1 **0**: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example , Cycle **800** and parameter **Reverse the tool Q498** = 0 Input: **0**, **1**
	Q531 Angle of incidence? Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**). Input: **-180...+180**

Example

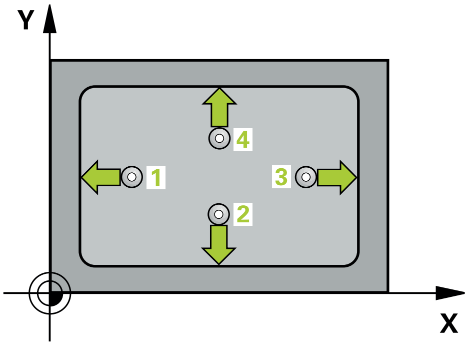
```
11 TCH PROBE 422 MEAS. CIRCLE OUTSIDE ~
    Q273=+50          ;CENTER IN 1ST AXIS ~
    Q274=+50          ;CENTER IN 2ND AXIS ~
    Q262=+75          ;NOMINAL DIAMETER ~
    Q325=+90          ;STARTING ANGLE ~
    Q247=+30          ;STEPPING ANGLE ~
    Q261=-5           ;MEASURING HEIGHT ~
    Q320=+0           ;SET-UP CLEARANCE ~
    Q260=+10          ;CLEARANCE HEIGHT ~
    Q301=+0           ;MOVE TO CLEARANCE ~
    Q277=+35.15       ;MAXIMUM LIMIT ~
    Q278=+34.9        ;MINIMUM LIMIT ~
    Q279=+0.05        ;TOLERANCE 1ST CENTER ~
    Q280=+0.05        ;TOLERANCE 2ND CENTER ~
    Q281=+1           ;MEASURING LOG ~
    Q309=+0           ;PGM STOP TOLERANCE ~
    Q330=+0           ;TOOL ~
    Q423=+4           ;NO. OF PROBE POINTS ~
    Q365=+1           ;TYPE OF TRAVERSE ~
    Q498=+0           ;REVERSE TOOL ~
    Q531=+0           ;ANGLE OF INCIDENCE
```

29.4.7 Cycle 423 MEAS. RECTAN. INSIDE

Application

Touch probe cycle **423** finds the center, length, and width of a rectangular pocket. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.

4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.

5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

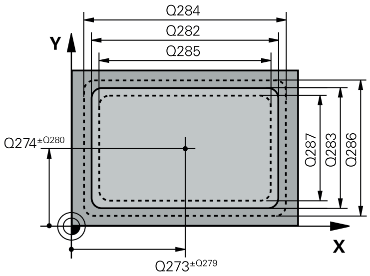
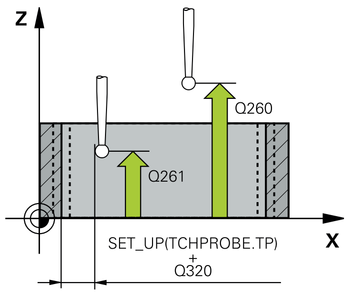
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q273 Center in 1st axis (nom. value)?** Center of the pocket in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q274 Center in 2nd axis (nom. value)? Center of the pocket in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q282 1st side length (nominal value)? Pocket length, parallel to the main axis of the working plane Input: **0...99999.9999**
	Q283 2nd side length (nominal value)? Pocket length, parallel to the secondary axis of the working plane Input: **0...99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q284 Max. size limit 1st side length? Maximum permissible length for the pocket Input: **0...99999.9999**
	Q285 Min. size limit 1st side length? Minimum permissible length for the pocket Input: **0...99999.9999**

Help graphic	Parameter
	Q286 Max. size limit 2nd side length? Maximum permissible width for the pocket Input: **0...99999.9999**
	Q287 Min. size limit 2nd side length? Minimum permissible width for the pocket Input: **0...99999.9999**
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: **0...99999.9999**
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log. **1**: Create a measuring log: The control will save the **log file named TCHPR423.TXT** in the folder that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen.Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control will monitor the tool Page 1609. **0**: Monitoring not active **> 0**: Tool number in tool table TOOL.T Input: **0...99999.9** or max. **255** characters

Example

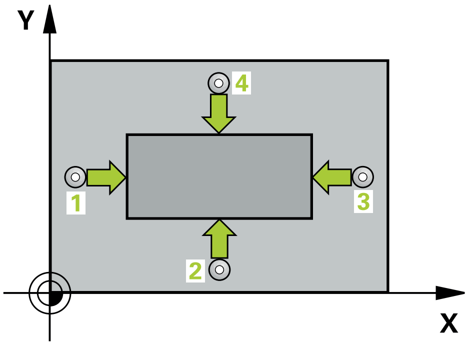
11 TCH PROBE 423 MEAS. RECTAN. INSIDE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q282=+80	;FIRST SIDE LENGTH ~
Q283=+60	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+10	;CLEARANCE HEIGHT ~
Q301=+1	;MOVE TO CLEARANCE ~
Q284=+0	;MAX. LIMIT 1ST SIDE ~
Q285=+0	;MIN. LIMIT 1ST SIDE ~
Q286=+0	;MAX. LIMIT 2ND SIDE ~
Q287=+0	;MIN. LIMIT 2ND SIDE ~
Q279=+0	;TOLERANCE 1ST CENTER ~
Q280=+0	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL

29.4.8 Cycle 424 MEAS. RECTAN. OUTS.

Application

Touch probe cycle **424** finds the center, length, and width of a rectangular stud.
If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

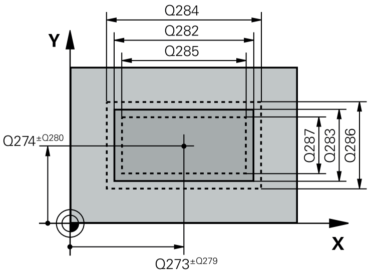
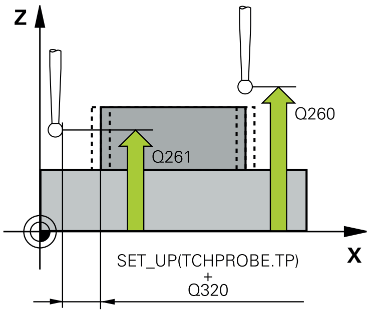
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q273 Center in 1st axis (nom. value)?** Center of the stud in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q274 Center in 2nd axis (nom. value)? Center of the stud in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q282 1st side length (nominal value)? Length of stud parallel to the main axis of the working plane Input: **0...99999.9999**
	Q283 2nd side length (nominal value)? Length of stud parallel to the secondary axis of the working plane Input: **0...99999.9999**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q284 Max. size limit 1st side length? Maximum permissible length for the stud Input: **0...99999.9999**
	Q285 Min. size limit 1st side length? Minimum permissible length for the stud Input: **0...99999.9999**

Help graphic	Parameter
	Q286 Max. size limit 2nd side length? Maximum permissible width for the stud Input: **0...99999.9999**
	Q287 Min. size limit 2nd side length? Minimum permissible width for the stud Input: **0...99999.9999**
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: **0...99999.9999**
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR424.TXT** in the folder that also contains the .h file **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1** **Q330 Tool for monitoring?** Define whether the control will monitor the tool Page 1609: **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters

Example

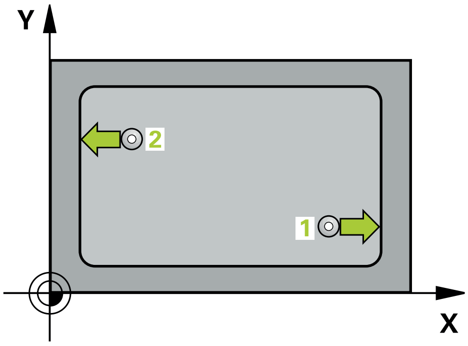
```
11 TCH PROBE 424 MEAS. RECTAN. OUTS. ~
    Q273=+50              ;CENTER IN 1ST AXIS ~
    Q274=+50              ;2ND CENTER 2ND AXIS ~
    Q282=+75              ;FIRST SIDE LENGTH ~
    Q283=+35              ;2ND SIDE LENGTH ~
    Q261=-5               ;MEASURING HEIGHT ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q260=+20              ;CLEARANCE HEIGHT ~
    Q301=+0               ;MOVE TO CLEARANCE ~
    Q284=+75.1            ;MAX. LIMIT 1ST SIDE ~
    Q285=+74.9            ;MIN. LIMIT 1ST SIDE ~
    Q286=+35              ;MAX. LIMIT 2ND SIDE ~
    Q287=+34.95           ;MIN. LIMIT 2ND SIDE ~
    Q279=+0.1             ;TOLERANCE 1ST CENTER ~
    Q280=+0.1             ;TOLERANCE 2ND CENTER ~
    Q281=+1               ;MEASURING LOG ~
    Q309=+0               ;PGM STOP TOLERANCE ~
    Q330=+0               ;TOOL
```

29.4.9 Cycle 425 MEASURE INSIDE WIDTH

Application

Touch probe cycle **425** measures the position and width of a slot (or pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation value in a Q parameter.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456
2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the positive direction of the programmed axis.
3 If you enter an offset for the second measurement, the control then moves the touch probe (if required, at clearance height) to the next touch point **2** and probes that point. If the nominal length is large, the control moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the control measures the width in the exact opposite direction.
4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Notes

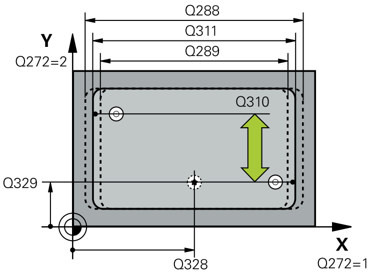
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter



Q328 Starting point in 1st axis?

Starting point for probing in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q329 Starting point in 2nd axis?

Starting point for probing in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q310 Offset for 2nd measuremnt (+/-)?

Distance by which the touch probe is offset before the second measurement. If you enter 0, the control does not offset the touch probe. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

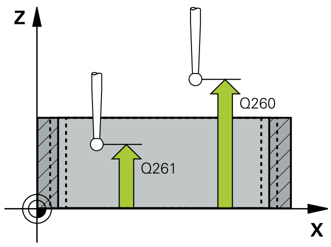
Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

1: Main axis = measuring axis

2: Secondary axis = measuring axis

Input: **1**, **2**



Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q311 Nominal length?

Nominal value of the length to be measured

Input: **0...99999.9999**

Q288 Maximum limit of size?

Maximum permissible length

Input: **0...99999.9999**

Q289 Minimum limit of size?

Minimum permissible length

Input: **0...99999.9999**

Help graphic	Parameter
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR425.TXT** in the folder that also contains the .h file **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control will monitor the tool Page 1609: **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**

Example

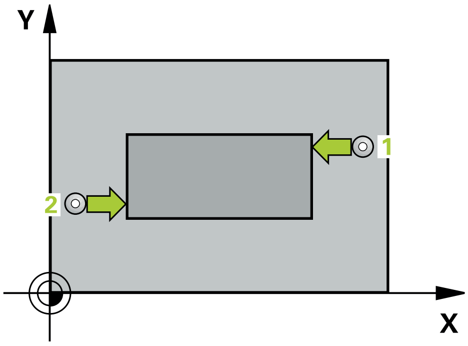
```
11 TCH PROBE 425 MEASURE INSIDE WIDTH ~
    Q328=+75              ;STARTNG PNT 1ST AXIS ~
    Q329=-12.5            ;STARTNG PNT 2ND AXIS ~
    Q310=+0               ;OFFS. 2ND MEASUREMNT ~
    Q272=+1               ;MEASURING AXIS ~
    Q261=-5               ;MEASURING HEIGHT ~
    Q260=+10              ;CLEARANCE HEIGHT ~
    Q311=+25              ;NOMINAL LENGTH ~
    Q288=+25.05           ;MAXIMUM LIMIT ~
    Q289=+25              ;MINIMUM LIMIT ~
    Q281=+1               ;MEASURING LOG ~
    Q309=+0               ;PGM STOP TOLERANCE ~
    Q330=+0               ;TOOL ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q301=+0               ;MOVE TO CLEARANCE
```

29.4.10 Cycle 426 MEASURE RIDGE WIDTH

Application

Touch probe cycle **426** measures the position and width of a ridge. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.

   Further information: "Positioning logic", Page 1456

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the negative direction of the programmed axis.
3 Then the touch probe moves at clearance height to the next touch point and probes it.
4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Notes

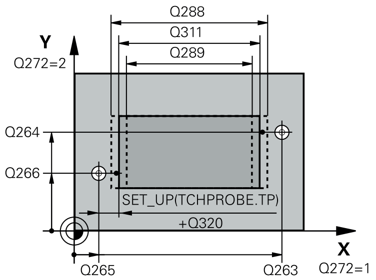
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic | **Parameter**



Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

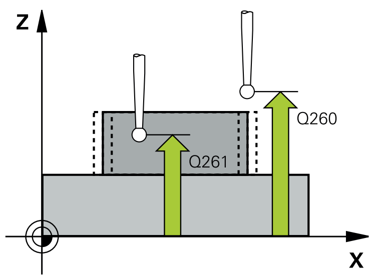
Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

1: Main axis = measuring axis

2: Secondary axis = measuring axis

Input: **1**, **2**



Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Q311 Nominal length?

Nominal value of the length to be measured

Input: **0...99999.9999**

Q288 Maximum limit of size?

Maximum permissible length

Input: **0...99999.9999**

Help graphic	Parameter
	Q289 Minimum limit of size? Minimum permissible length Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR426.TXT** in the folder that also contains the associated NC program. **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control will monitor the tool Page 1609: **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters

Example

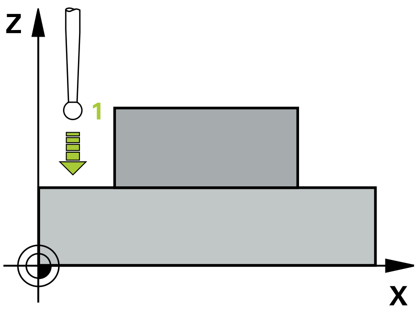
```
11 TCH PROBE 426 MEASURE RIDGE WIDTH ~
    Q263=+50              ;1ST POINT 1ST AXIS ~
    Q264=+25              ;1ST POINT 2ND AXIS ~
    Q265=+50              ;2ND PNT IN 1ST AXIS ~
    Q266=+85              ;2ND PNT IN 2ND AXIS ~
    Q272=+2               ;MEASURING AXIS ~
    Q261=-5               ;MEASURING HEIGHT ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q260=+20              ;CLEARANCE HEIGHT ~
    Q311=+45              ;NOMINAL LENGTH ~
    Q288=+45              ;MAXIMUM LIMIT ~
    Q289=+44.95           ;MINIMUM LIMIT ~
    Q281=+1               ;MEASURING LOG ~
    Q309=+0               ;PGM STOP TOLERANCE ~
    Q330=+0               ;TOOL
```

29.4.11 Cycle 427 MEASURE COORDINATE

Application

Touch probe cycle **427** measures a coordinate in a selectable axis and saves the value in a Q parameter. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction

   Further information: "Positioning logic", Page 1456
2 Then the control positions the touch probe to the specified touch point **1** in the working plane and measures the actual value in the selected axis.
3 Finally, the control returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter:

Q parameter number	Meaning
Q160	Measured coordinate

Notes

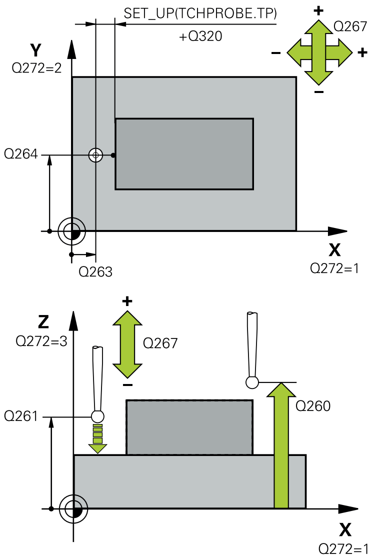
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If an axis of the active working plane is defined as the measuring axis (**Q272** = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing direction (**Q267**).
- If the touch probe axis is defined as the measuring axis (**Q272** = 3), the control will perform a tool length compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter Q330 references a turning tool, the following applies:
  - Parameters **Q498** and **Q531** have to be defined
  - The information in parameters **Q498**, **Q531**, for example from Cycle **800**, has to match this information
  - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
  - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** or **PREDEF**

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

1: Main axis = measuring axis

2: Secondary axis = measuring axis

3: Touch probe axis = measuring axis

Input: **1**, **2**, **3**

Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

-1: Negative traverse direction

+1: Positive traverse direction

Input: **-1**, **+1**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.

Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR427.TXT** in the folder that also contains the associated NC program. **2:** Interrupt the program run and display the measuring log on the control screen.Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q288 Maximum limit of size? Maximum permissible value Input: **-99999.9999...+99999.9999**
	Q289 Minimum limit of size? Minimum permissible value Input: **-99999.9999...+99999.9999**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control will monitor the tool Page 1609: **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters

Help graphic	Parameter
	Q498 Reverse tool (0=no/1=yes)? Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following: **1**: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 1 **0**: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example , Cycle **800** and parameter **Reverse the tool Q498** = 0 Input: **0**, **1**
	Q531 Angle of incidence? Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**). Input: **-180...+180**

Example

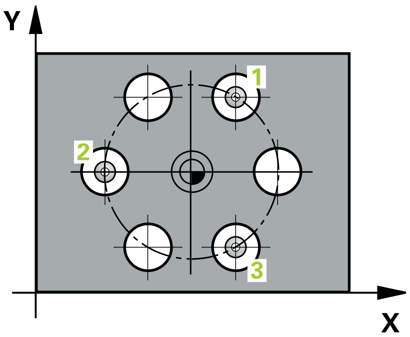
```
11 TCH PROBE 427 MEASURE COORDINATE ~
    Q263=+35            ;1ST POINT 1ST AXIS ~
    Q264=+45            ;1ST POINT 2ND AXIS ~
    Q261=+5             ;MEASURING HEIGHT ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q272=+3             ;MEASURING AXIS ~
    Q267=-1             ;TRAVERSE DIRECTION ~
    Q260=+20            ;CLEARANCE HEIGHT ~
    Q281=+1             ;MEASURING LOG ~
    Q288=+5.1           ;MAXIMUM LIMIT ~
    Q289=+4.95          ;MINIMUM LIMIT ~
    Q309=+0             ;PGM STOP TOLERANCE ~
    Q330=+0             ;TOOL ~
    Q498=+0             ;REVERSE TOOL ~
    Q531=+0             ;ANGLE OF INCIDENCE
```

29.4.12 Cycle 430 MEAS. BOLT HOLE CIRC

Application

Touch probe cycle **430** finds the center and diameter of a bolt hole circle by probing three holes. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed center point of the first hole **1**.

   Further information: "Positioning logic", Page 1456
2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
7 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

Q parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation of bolt circle diameter

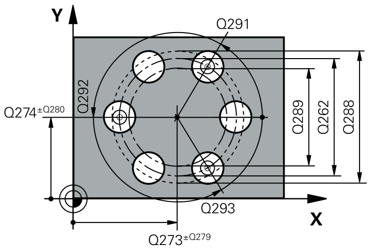
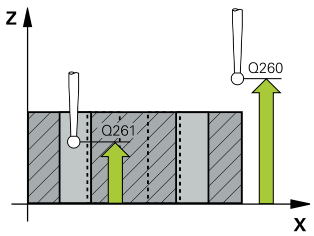
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **430** only monitors for tool breakage; there is no automatic tool compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q273 Center in 1st axis (nom. value)?** Bolt hole circle center (nominal value) in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q274 Center in 2nd axis (nom. value)? Bolt hole circle center (nominal value) in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q262 Nominal diameter? Enter the diameter of the hole. Input: **0...99999.9999**
	Q291 Polar coord. angle of 1st hole? Polar coordinate angle of the first hole center in the working plane. The value has an absolute effect. Input: **-360.000...+360.000**
	Q292 Polar coord. angle of 2nd hole? Polar coordinate angle of the second hole center in the working plane. The value has an absolute effect. Input: **-360.000...+360.000**
	Q293 Polar coord. angle of 3rd hole? Polar coordinate angle of the third hole center in the working plane. The value has an absolute effect. Input: **-360.000...+360.000**
	**Q261 Measuring height in probe axis?** Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q288 Maximum limit of size? Maximum permissible diameter of bolt hole circle Input: **0...99999.9999**
	Q289 Minimum limit of size? Minimum permissible diameter of bolt hole circle Input: **0...99999.9999**
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: **0...99999.9999**

Help graphic	Parameter
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: **0...99999.9999**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR430.TXT** in the folder that also contains the associated NC program **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run; no error message **1**: Interrupt program run and output an error message Input: **0**, **1**
	Q330 Tool for monitoring? Define whether the control will monitor the tool Page 1609: **0**: Monitoring not active **> 0**: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: **0...99999.9** or max. **255** characters

Example

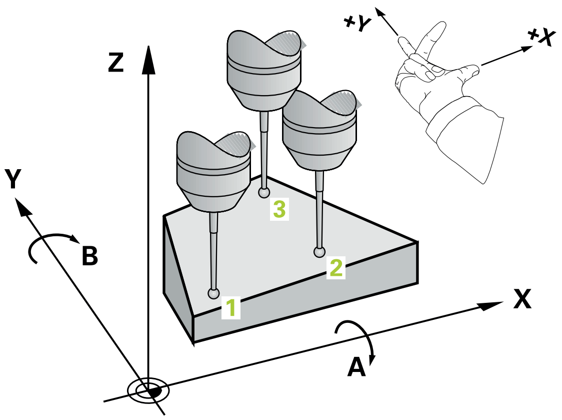
```
11 TCH PROBE 430 MEAS. BOLT HOLE CIRC ~
    Q273=+50            ;CENTER IN 1ST AXIS ~
    Q274=+50            ;CENTER IN 2ND AXIS ~
    Q262=+80            ;NOMINAL DIAMETER ~
    Q291=+0             ;ANGLE OF 1ST HOLE ~
    Q292=+90            ;ANGLE OF 2ND HOLE ~
    Q293=+180           ;ANGLE OF 3RD HOLE ~
    Q261=-5             ;MEASURING HEIGHT ~
    Q260=+10            ;CLEARANCE HEIGHT ~
    Q288=+80.1          ;MAXIMUM LIMIT ~
    Q289=+79.9          ;MINIMUM LIMIT ~
    Q279=+0.15          ;TOLERANCE 1ST CENTER ~
    Q280=+0.15          ;TOLERANCE 2ND CENTER ~
    Q281=+1             ;MEASURING LOG ~
    Q309=+0             ;PGM STOP TOLERANCE ~
    Q330=+0             ;TOOL
```

29.4.13 Cycle 431 MEASURE PLANE

Application

Touch probe cycle **431** finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

Cycle sequence



1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) at the programmed touch point **1** and measures the first point of the plane. The control offsets the touch probe by the set-up clearance in the direction opposite to the direction of probing.

   Further information: "Positioning logic", Page 1456
2 The touch probe returns to the clearance height and then moves in the working plane to touch point **2** and measures the actual value of the second touch point in the plane.
3 The touch probe returns to the clearance height and then moves in the working plane to touch point **3** and measures the actual value of the third touch point in the plane.
4 Finally the control returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

Q parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to **Q175**	Measured values in the touch probe axis (first to third measurement)

Notes

NOTICE

Danger of collision!

If you save the angle values in the preset table and then tilt the tool by programming **PLANE SPATIAL** with **SPA** = 0; **SPB** = 0; **SPC** = 0, there are multiple solutions in which the tilting axes are at 0.

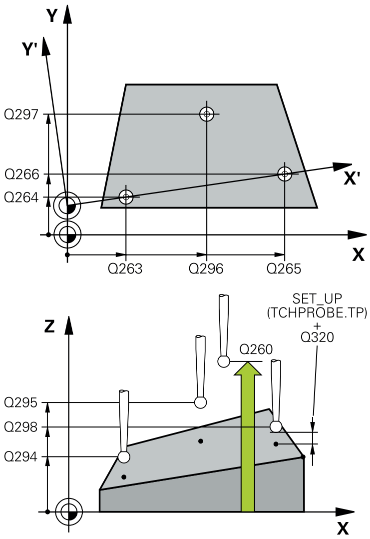
- Make sure to program **SYM** (**SEQ**) + or **SYM** (**SEQ**) -

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three measuring points are not positioned on a straight line.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The spatial angles that are needed for the **Tilt working plane** function are saved in parameters **Q170** to **Q172**. With the first two measuring points, you also specify the direction of the main axis when tilting the working plane.
- The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

Cycle parameters

Help graphic	Parameter
	**Q263 1st measuring point in 1st axis?** Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q265 2nd measuring point in 1st axis? Coordinate of the second touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q266 2nd measuring point in 2nd axis? Coordinate of the second touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q295 2nd measuring point in 3rd axis? Coordinate of the second touch point in the touch probe axis. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q296 3rd measuring point in 1st axis? Coordinate of the third touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q297 3rd measuring point in 2nd axis? Coordinate of the third touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q298 3rd measuring point in 3rd axis? Coordinate of the third touch point in the touch probe axis. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**

Help graphic	Parameter
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: **0**: Do not create a measuring log **1**: Create a measuring log: The control will save the **log file named TCHPR431.TXT** in the folder that also contains the associated NC program **2**: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**. Input: **0**, **1**, **2**

Example

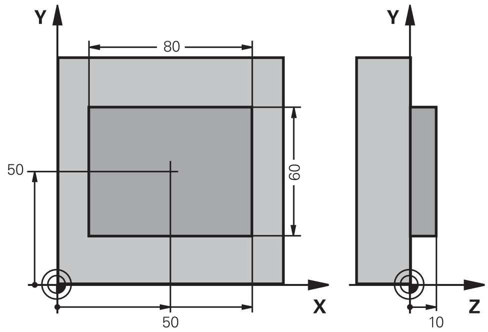
```
11 TCH PROBE 431 MEASURE PLANE ~
    Q263=+20              ;1ST POINT 1ST AXIS ~
    Q264=+20              ;1ST POINT 2ND AXIS ~
    Q294=-10              ;1ST POINT 3RD AXIS ~
    Q265=+50              ;2ND PNT IN 1ST AXIS ~
    Q266=+80              ;2ND PNT IN 2ND AXIS ~
    Q295=+0               ;2ND PNT IN 3RD AXIS ~
    Q296=+90              ;3RD PNT IN 1ST AXIS ~
    Q297=+35              ;THIRD POINT 2ND AXIS ~
    Q298=+12              ;3RD PNT IN 3RD AXIS ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q260=+5               ;CLEARANCE HEIGHT ~
    Q281=+1               ;MEASURING LOG
```

29.4.14 Programming Examples

Example: Measuring and reworking a rectangular stud

Program sequence

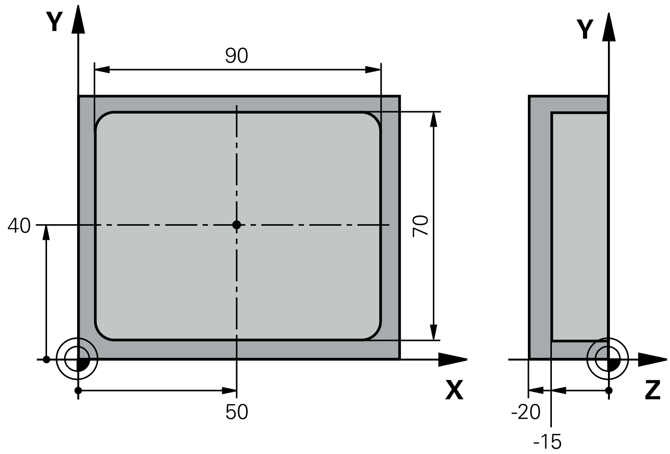
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



0 BEGIN PGM TOUCHPROBE MM	
1 TOOL CALL 5 Z S6000	; Tool call: roughing
2 Q1 = 81	; Rectangle length in X (roughing dimension)
3 Q2 = 61	; Rectangle length in Y (roughing dimension)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CALL LBL 1	; Call the subprogram for machining
6 L Z+100 R0 FMAX	; Retract the tool
7 TOOL CALL 600 Z	; Call the touch probe
8 TCH PROBE 424 MEAS. RECTAN. OUTS. ~	
Q273=+50 ;CENTER IN 1ST AXIS ~	
Q274=+50 ;CENTER IN 2ND AXIS ~	
Q282=+80 ;FIRST SIDE LENGTH ~	
Q283=+60 ;2ND SIDE LENGTH ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q320=+0 ;SET-UP CLEARANCE ~	
Q260=+30 ;CLEARANCE HEIGHT ~	
Q301=+0 ;MOVE TO CLEARANCE ~	
Q284=+0 ;MAX. LIMIT 1ST SIDE ~	
Q285=+0 ;MIN. LIMIT 1ST SIDE ~	
Q286=+0 ;MAX. LIMIT 2ND SIDE ~	
Q287=+0 ;MIN. LIMIT 2ND SIDE ~	
Q279=+0 ;TOLERANCE 1ST CENTER ~	
Q280=+0 ;TOLERANCE 2ND CENTER ~	
Q281=+0 ;MEASURING LOG ~	
Q309=+0 ;PGM STOP TOLERANCE ~	
Q330=+0 ;TOOL	

9 Q1 = Q1 - Q164	; Calculate the length in X based on the measured deviation
10 Q2 = Q2 - Q165	; Calculate the length in Y based on the measured deviation
11 L Z+100 R0 FMAX	; Retract the touch probe
12 TOOL CALL 25 Z S8000	; Tool call: finishing
13 L Z+100 R0 FMAX M3	; Retract the tool, end of program
14 CALL LBL 1	; Call the subprogram for machining
15 L Z+100 R0 FMAX	
16 M30	
17 LBL 1	; Subprogram with rectangular stud machining cycle
18 CYCL DEF 256 RECTANGULAR STUD ~	
Q218=+Q1 ;FIRST SIDE LENGTH ~	
Q424=+82 ;WORKPC. BLANK SIDE 1 ~	
Q219=+Q2 ;2ND SIDE LENGTH ~	
Q425=+62 ;WORKPC. BLANK SIDE 2 ~	
Q220=+0 ;RADIUS / CHAMFER ~	
Q368=+0.1 ;ALLOWANCE FOR SIDE ~	
Q224=+0 ;ANGLE OF ROTATION ~	
Q367=+0 ;STUD POSITION ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-10 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q206=+3000 ;FEED RATE FOR PLNGNG ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+10 ;SURFACE COORDINATE ~	
Q204=+20 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q437=+0 ;APPROACH POSITION ~	
Q215=+0 ;MACHINING OPERATION ~	
Q369=+0 ;ALLOWANCE FOR FLOOR ~	
Q338=+20 ;INFEED FOR FINISHING ~	
Q385=+500 ;FINISHING FEED RATE	
19 L X+50 Y+50 R0 FMAX M99	; Cycle call
20 LBL 0	; End of subprogram
21 END PGM TOUCHPROBE MM	

Example: Measuring a rectangular pocket and recording the results



0 BEGIN PGM TOUCHPROBE_2 MM	
1 TOOL CALL 600 Z	; Tool call: touch probe
2 L Z+100 R0 FMAX	; Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE ~	
Q273=+50 ;CENTER IN 1ST AXIS ~	
Q274=+40 ;CENTER IN 2ND AXIS ~	
Q282=+90 ;FIRST SIDE LENGTH ~	
Q283=+70 ;2ND SIDE LENGTH ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q320=+2 ;SET-UP CLEARANCE ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q301=+0 ;MOVE TO CLEARANCE ~	
Q284=+90.15 ;MAX. LIMIT 1ST SIDE ~	
Q285=+89.95 ;MIN. LIMIT 1ST SIDE ~	
Q286=+70.1 ;MAX. LIMIT 2ND SIDE ~	
Q287=+69.9 ;MIN. LIMIT 2ND SIDE ~	
Q279=+0.15 ;TOLERANCE 1ST CENTER ~	
Q280=+0.1 ;TOLERANCE 2ND CENTER ~	
Q281=+1 ;MEASURING LOG ~	
Q309=+0 ;PGM STOP TOLERANCE ~	
Q330=+0 ;TOOL	
4 L Z+100 R0 FMAX	; Retract the tool, end of program
5 M30	
6 END PGM TOUCHPROBE_2 MM	

29.5 Touch Probe Cycles: Special Functions

29.5.1 Fundamentals

Overview

The control must be specifically prepared by the machine manufacturer for the use of a touch probe.

HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

The control provides cycles for the following special purposes:

Cycle		Call	Further information
3	**MEASURING** ▪ Touch probe cycle for defining OEM cycles	**DEF-**active	Page 1663
4	**MEASURING IN 3-D** ▪ Measuring any position	**DEF-**active	Page 1665
444	**PROBING IN 3-D** ▪ Measuring any position ▪ Determining the deviation from the nominal coordinates	**DEF-**active	Page 1668
441	**FAST PROBING** ▪ Touch probe cycle for defining various touch probe parameters	**DEF-**active	Page 1674
1493	**EXTRUSION PROBING** ▪ Touch probe cycle for defining an extrusion ▪ Extrusion direction, length, and number of extrusion points can be programmed	**DEF-**active	Page 1676

29.5.2 Cycle 3 MEASURING

Application

Touch probe cycle **3** measures any position on the workpiece in a selectable probing direction. Unlike other touch probe cycles, Cycle **3** enables you to enter the measuring range **SET UP** and feed rate **F** directly. Also, the touch probe retracts by a definable value **MB** after determining the measured value.

Cycle sequence

1 The touch probe moves from the current position at the specified feed rate in the defined probing direction. Use polar angles to define the probing direction in the cycle.
2 After the control has saved the position, the touch probe stops. The control saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. The control does not conduct any length or radius compensations. You define the number of the first result parameter in the cycle.
3 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

Notes

> The exact behavior of touch probe cycle **3** is defined by your machine tool builder or a software manufacturer who uses it within specific touch probe cycles.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) touch-probe data, which are effective in other touch probe cycles, do not apply in touch probe cycle **3**.
- Remember that the control always writes to four successive Q parameters.
- If the control was not able to determine a valid touch point, the NC program is run without error message. In this case the control assigns the value –1 to the fourth result parameter so that you can deal with the error yourself.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.

> With function **FN17: SYSWRITE ID 990 NR 6** you can set whether the cycle sequences through the probe input X12 or X13.

Cycle parameters

Help graphic	Parameter
	Parameter number for result? Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters. Input: **0...1999**
	Probing axis? Enter the axis in whose direction the touch probe will move and confirm with the **ENT** key. Input: **X**, **Y**, or **Z**
	Probing angle? Angle measured from the defined **probing axis** in which the touch probe will move. Confirm with **ENT**. Input: **-180...+180**
	Maximum measuring range? Enter the maximum distance from the starting point by which the touch probe will move. Confirm with ENT. Input: **-999999999...+999999999**
	Feed rate measurement Enter the measuring feed rate in mm/min. Input: **0...3000**
	Maximum retraction distance? Traverse path in the direction opposite the probing direction, after the stylus was deflected. The control returns the touch probe to a point no farther than the starting point, so that there can be no collision. Input: **0...999999999**
	Reference system? (0=ACT/1=REF) Define whether the probing direction and measurement result will be referenced to the current coordinate system (**ACT**, can be shifted or rotated) or the machine coordinate system (**REF**): **0**: Perform the probing operation in the current system and save the measurement result in the **ACT** system **1**: Perform the probing operation in the machine-based REF system. Save the measurement result in the REF system. Input: **0**, **1**

Help graphic	Parameter
	Error mode? (0=OFF/1=ON) Define whether the control will issue an error message if the stylus is deflected at cycle start. If mode **1** is selected, the control saves the value **-1** in the 4th result parameter and continues the cycle: **0**: Issue error message **1**: Do not issue error message Input: **0**, **1**

Example

```
11 TCH PROBE 3.0 MEASURING
12 TCH PROBE 3.1 Q1
13 TCH PROBE 3.2 X ANGLE:+15
14 TCH PROBE 3.3 ABST+10 F100 MB1 REFERENCE SYSTEM:0
15 TCH PROBE 3.4 ERRORMODE1
```

29.5.3 Cycle 4 MEASURING IN 3-D

Application

Touch probe cycle **4** measures any position on the workpiece in the probing direction defined by a vector. Unlike other touch probe cycles, Cycle **4** enables you to enter the probing distance and probing feed rate directly. You can also define the distance by which the touch probe retracts after acquiring the probed value.

Cycle **4** is an auxiliary cycle that can be used for probing with any touch probe (TS or TT). The control does not provide a cycle for calibrating the TS touch probe in any probing direction.

Cycle sequence

1 The control moves the touch probe from the current position at the entered feed rate in the defined probing direction. Define the probing direction in the cycle by using a vector (delta values in X, Y and Z).

2 After the control has saved the position, the control stops the probe movement. The control saves the X, Y, Z coordinates of the probing position in three successive Q parameters. You define the number of the first parameter in the cycle. If you are using a TS touch probe, the probe result is corrected by the calibrated center offset.

3 Finally, the control retracts the touch probe in the direction opposite to the direction of probing. You define the traverse distance in parameter **MB**—the touch probe is moved to a point no farther than the starting point.

> Ensure during pre-positioning that the control moves the probe-tip center without compensation to the defined position.

Notes

NOTICE
Danger of collision! If the control was not able to determine a valid touch point, the 4th result parameter will have the value –1. The control does **not** interrupt the program run! ▶ Make sure that all touch points can be reached.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.
- Remember that the control always writes to four successive Q parameters.

Cycle parameters

Help graphic	Parameter
	Parameter number for result? Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters. Input: **0...1999**
	Relative measuring path in X? X component of the direction vector defining the direction in which the touch probe will move. Input: **-999999999...+999999999**
	Relative measuring path in Y? Y component of the direction vector defining the direction in which the touch probe will move. Input: **-999999999...+999999999**
	Relative measuring path in Z? Z component of the direction vector defining the direction in which the touch probe will move. Input: **-999999999...+999999999**
	Maximum measuring range? Enter the maximum distance from the starting point by which the touch probe will move along the direction vector. Input: **-999999999...+999999999**
	Feed rate measurement Enter the measuring feed rate in mm/min. Input: **0...3000**
	Maximum retraction distance? Traverse path in the direction opposite the probing direction, after the stylus was deflected. Input: **0...999999999**
	Reference system? (0=ACT/1=REF) Define whether the result of probing will be saved in the input coordinate system (**ACT**), or with respect to the machine coordinate system (**REF**): **0**: Save the measurement result in the **ACT** system **1**: Save the measurement result in the **REF** system Input: **0**, **1**

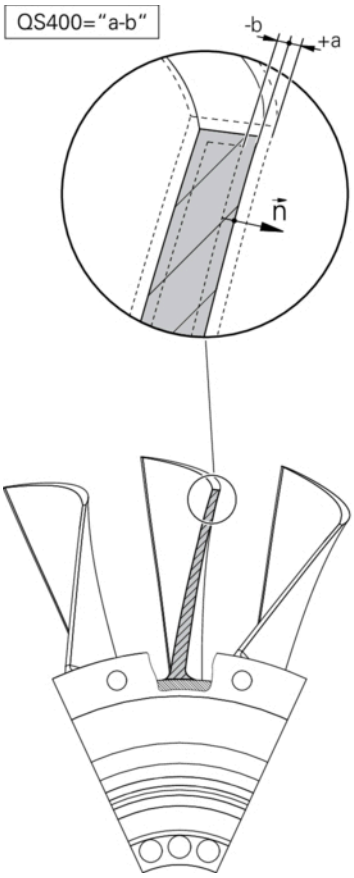
Example

```
11 TCH PROBE 4.0 MEASURING IN 3-D
12 TCH PROBE 4.1 Q1
13 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1
14 TCH PROBE 4.3 ABST+45 F100 MB50 REFERENCE SYSTEM:0
```

29.5.4 Cycle 444 PROBING IN 3-D

Application

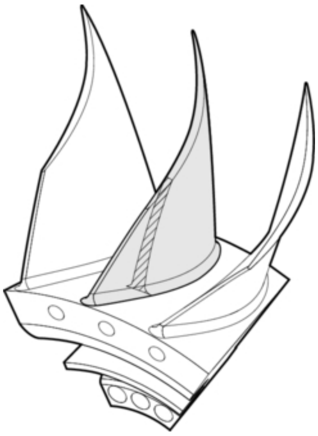
> Refer to your machine manual.
> This function must be enabled and adapted by the machine manufacturer.



Cycle **444** checks one specific point on the surface of a component. This cycle is used, for example, to measure free-form surfaces of molded parts. It can be determined whether a point on the surface of the component lies in an undersize or oversize range compared to a nominal coordinate. The operator can subsequently perform further machining steps, such as reworking.

Cycle **444** probes any point in three dimensions, and determines the deviation from a nominal coordinate. A normal vector, defined in parameters **Q581**, **Q582**, and **Q583**, is used for this purpose. The normal vector is perpendicular to an imagined surface in which the nominal coordinate is located. The normal vector points away from the surface, and does not determine the probing path. It is advisable to determine the normal vector with the help of a CAD or CAM system. A tolerance range **QS400** defines the permissible deviation between the actual and nominal coordinate along the normal vector. This way you define, for example, that the program is to be interrupted if an undersize is detected. Additionally, the control outputs a log and the deviations are stored in the Q parameters listed below.

Cycle sequence



1 Starting from the current position, the touch probe traverses to a point on the normal vector that is at the following distance from the nominal coordinate: Distance = ball-tip radius + **SET_UP** value from the tchprobe.tp table (TNC:\table \tchprobe.tp) + **Q320**. Pre-positioning takes a clearance height into account.Page 1456
2 The touch probe then approaches the nominal coordinate. The probing distance is defined by DIST, not by the normal vector! The normal vector is only used for the correct calculation of the coordinates.
3 After the control has saved the position, the touch probe is retracted and stopped. The control saves the measured coordinates of the contact point in Q parameters.
4 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

Result parameters

The control stores the probing results in the following parameters:

Q parameter number	Meaning
Q151	Measured position in main axis
Q152	Measured position in secondary axis
Q153	Measured position in tool axis
Q161	Measured deviation in main axis
Q162	Measured deviation in secondary axis
Q163	Measured deviation in tool axis
Q164	Measured 3-D deviation ■ Less than 0: Undersize ■ Greater than 0: Oversize
Q183	Workpiece status: ■ - 1 = undefined ■ 0 = good ■ 1 = Rework ■ 2 = Scrap

Log function

Once probing has finished, the control generates a log in HTML format. The log includes the results from the main, secondary, and tool axes as well as the 3-D deviation. The control saves the log in the same folder in which the *.h file is located (as long as no path has been configured for FN16).

The log contains the following data on the main, secondary, and tool axes:

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- If a tolerance **QS400** was defined: Upper and lower dimensions are output, as well as the determined deviation along the normal vector
- Ascertained actual coordinate
- Colored display of the values (green for "good," orange for "rework," red for "scrap")

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- In order to obtain exact results from the touch probe being used, you need to perform 3-D calibration before executing Cycle **444**. Software option 92, **3D#ToolComp**, is required for 3-D calibration.
- Cycle **444** generates a measuring log in HTML format.
- An error message is output if Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, or Cycle **26 AXIS-SPECIFIC SCALING** is active before Cycle **444** is run.
- For probing, an active TCPM will be taken into account. While the TCPM is active, probing of positions is possible even if the position resulting from the **Tilt working plane** function is inconsistent with the current position of the rotary axes.
- If your machine is equipped with a feedback-controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.
- Cycle **444** references all coordinates to the input system.
- The control writes the measured values to return parameters, Page 1668.
- The workpiece status (good/rework/scrap) is set via Q parameter **Q183**, independent of parameter **Q309** (Page 1668).

Note regarding machine parameters

- Depending on the setting of the optional machine parameter **chkTiltingAxes** (no. 204600), the control will check during probing whether the position of the rotary axes matches the tilting angles (3D-ROT). If that is not the case, the control displays an error message.

Cycle parameters

Help graphic	Parameter
	Q263 1st measuring point in 1st axis? Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. The value has an absolute effect. Input: **-99999.9999...+99999.9999**
	Q581 Surface-normal in ref. axis? Enter here the surface normal in the direction of the main axis. The surface normal of a point is normally output by a CAD/CAM system. Input: **-10...+10**
	Q582 Surface-normal in minor axis? Enter here the surface normal in the direction of the secondary axis. The surface normal of a point is normally output by a CAD/CAM system. Input: **-10...+10**
	Q583 Surface-normal in tool axis? Enter here the surface normal in the direction of the tool axis. The surface normal of a point is normally output by a CAD/CAM system. Input: **-10...+10**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: **-99999.9999...+99999.9999** or **PREDEF**

Help graphic	Parameter
	QS400 Tolerance value? Specify a tolerance band that will be monitored by the cycle. The tolerance defines the deviation permitted along the surface normal. This deviation is determined between the nominal coordinate and the actual coordinate of the workpiece. (The surface normal is defined by **Q581** to **Q583**, and the nominal coordinate is defined by **Q263**, **Q264**, and **Q294**.) The tolerance value is distributed over the axes, depending on the normal vector (see examples). **Examples** ■ **QS400 ="0.4-0.1"** means: Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate –0.1. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate –0.1" ■ **QS400 ="0.4"** means: Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate". ■ **QS400 ="-0.1"** means: Upper dimension = nominal coordinate; lower dimension = nominal coordinate –0.1. The following tolerance band thus results for the cycle: "nominal coordinate" to "nominal coordinate -0.1". ■ **QS400 =" "** means: No tolerance band. ■ **QS400 ="0"** means: No tolerance band. ■ **QS400 ="0.1+0.1"** means: No tolerance band. Input: Max. **255** characters
	Q309 Reaction to tolerance error? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: **0**: Do not interrupt program run when tolerance is exceeded; do not output an error message **1**: Interrupt program run when tolerance is exceeded and output an error message **2:** If the value of the measured actual coordinate along the surface normal vector is less than the nominal coordinate, the control displays a message and interrupts the NC program run. However, there will be no error message if the value of the measured actual coordinate is greater than the nominal coordinate. Input: **0**, **1**, **2**

Example

11 TCH PROBE 444 PROBING IN 3-D ~	
Q263=+0	;1ST POINT 1ST AXIS ~
Q264=+0	;1ST POINT 2ND AXIS ~
Q294=+0	;1ST POINT 3RD AXIS ~
Q581=+1	;NORMAL IN REF. AXIS ~
Q582=+0	;NORMAL IN MINOR AXIS ~
Q583=+0	;NORMAL IN TOOL AXIS ~
Q320=+0	;SAFETY CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
QS400="1-1"	;TOLERANCE ~
Q309=+0	;ERROR REACTION

29.5.5 Cycle 441 FAST PROBING

Application

You can use touch probe cycle **441** to globally specify various touch probe parameters (e.g. the positioning feed rate) for all subsequently used touch probe cycles.

The purpose of Cycle **441** is to set parameters for probing cycles. In this cycle, no machine movements will be performed.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- **END PGM**, **M2**, **M30** reset the global settings of Cycle **441**.
- Cycle parameter **Q399** depends on your machine configuration. Your machine tool builder is responsible for the setting of whether the touch probe can be oriented through an NC program.
- Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate with the feed rate potentiometer only, even with **Q397**=1.

Note regarding machine parameters

- The machine parameter **maxTouchFeed** (no. 122602) allows the machine manufacturer to limit the feed rate. You define the maximum absolute feed rate in this machine parameter.

Cycle parameters

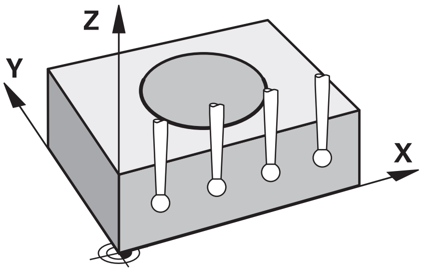
Help graphic	Parameter
	Q396 Positioning feed rate? Define the feed rate at which the touch probe will be moved to the specified positions. Input: **0...99999.999**
	Q397 Pre-pos. at machine's rapid? Define whether the control, when prepositioning the touch probe, traverses at **FMAX** feed rate (machine's rapid traverse): **0**: Pre-position at the feed rate from **Q396** **1**: Pre-position at the machine's rapid traverse **FMAX** Input: **0**, **1**
	Q399 Angle tracking (0/1)? Define whether the control will orient the touch probe before every probing operation: **0**: Do not orient the spindle **1**: Orient the spindle before every probing operation (increased accuracy)? Input: **0**, **1**
	Q400 Automatic interruption? Define whether the control will interrupt program run and output the measurement results on the screen following a touch probe cycle for automatic workpiece measurement: **0**: Do not interrupt program run even if, in the specific touch probe cycle, the output of measurement results on the screen is selected **1**: Interrupt program run and output measurement results on the screen. You can then resume the NC program run with **NC Start**. Input: **0**, **1**

Example

```
11 TCH PROBE 441 FAST PROBING ~
    Q396=+3000          ;POSITIONING FEEDRATE ~
    Q397=+0             ;SELECT FEED RATE ~
    Q399=+1             ;ANGLE TRACKING ~
    Q400=+1             ;INTERRUPTION
```

29.5.6 Cycle 1493 EXTRUSION PROBING

Application



Cycle **1493** allows you to repeat the touch points of specific touch probe cycles along a straight line. In the cycle, you define the direction and the length of the extrusion, as well as the number of extrusion points.

The repetitions allow you, for example, to perform multiple measurements at different heights and to determine deviations based on the deflection of the tool. You can also use the extrusion to increase the accuracy during probing. Multiple measuring points help you ascertain contamination on the workpiece or rough surfaces.

In order to activate the repetition of specific touch points, you need to define Cycle **1493** before the probing cycle. Depending on the definition, this cycle will remain active for only the next cycle or for the entire NC program. The control interprets the extrusion in the input coordinate system **I-CS**.

The following cycles are capable of performing extrusions:

- **PROBING IN PLANE** (Cycle **1420**, option 17), see Page 1472
- **PROBING ON EDGE** (Cycle **1410**), see Page 1478
- **PROBING TWO CIRCLES** (Cycle **1411**), see Page 1485
- **INCLINED EDGE PROBING** (Cycle **1412**), see Page 1493
- **POSITION PROBING** (Cycle **1400**), see Page 1525
- **CIRCLE PROBING** (Cycle **1401**), see Page 1529

Result parameters

The control stores the results of the probing cycle in the following Q parameters:

Q parameter number	Meaning
Q970	Maximum deviation from the ideal line of touch point 1
Q971	Maximum deviation from the ideal line of touch point 2
Q972	Maximum deviation from the ideal line of touch point 3
Q973	Maximum deviation of diameter 1
Q974	Maximum deviation of diameter 2

QS parameter

In addition to saving the results of the probing cycle in the return parameters **Q97x**, the control saves individual results to the QS parameters **QS97x**. The control saves the results of all measuring points from **one** extrusion in the corresponding QS parameters. Each result is ten characters long and the results are separated from each other by a blank space. This makes it easy for the control to convert the individual values in the NC program via string processing and use them for special automated evaluations.

Result in a QS parameter:

QS970 = "0.12345678 -1.1234567 -2.1234567 -3.12345678"

Further information: "String functions", Page 1301

Log function

Once probing has finished, the control generates a log file in HTML format. The log file contains the results of the 3-D deviation in graphical and tabular form. The control saves the log file in the same folder in which the NC program is located.

The log file contains the following data in the main axis, secondary axis and tool axis depending on the selected cycle (e.g., circle center point and diameter):

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- Upper and lower dimensions, as well as the determined deviation along the normal vector
- Measured actual coordinate
- Color coding of the values:
  - Green: Good
  - Orange: Rework
  - Red: Scrap
- Extrusion points

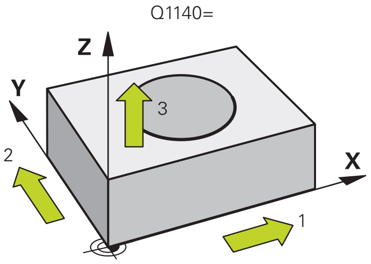
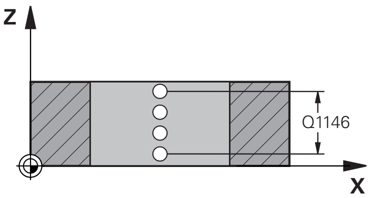
Extrusion points:

The horizontal axis represents the direction for the extrusion. The blue points are the individual measuring points. The red lines indicate the lower limit and the upper limit of the dimensions. If a value violates a specified tolerance, the control will show the area in red color in the graphic.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If **Q1145 > 0** and **Q1146 = 0**, then the control will perform the number of extrusion points at the same position.
- If you use Cycle **1401 CIRCLE PROBING** or **1411 PROBING TWO CIRCLES** to perform an extrusion, the direction for the extrusion must be **Q1140 = +3**; otherwise, the control will output an error message.

Cycle parameters

Help graphic	Parameter
	**Q1140 Direction for extrusion (1-3)?** **1:** Extrusion in the direction of the main axis **2:** Extrusion in the direction of the secondary axis **3:** Extrusion in the direction of the tool axis Input: **1**, **2**, **3**
	Q1145 Number of extrusion points? Number of measuring points that the cycle repeats over the length of the extrusion **Q1146**. Input: **1...99**
	**Q1146 Length of extrusion?** Length over which the measuring points are repeated. Input: **-99...+99**
	Q1149 Extrusion: Modal duration? Effect of the cycle: **0**: The extrusion is effective for only the next cycle. **1:** The extrusion is effective until the end of the NC program. Input: **-99...+99**

Example

```
11 TCH PROBE 1493 EXTRUSION PROBING ~
    Q1140=+3                ;EXTRUSION DIRECTION ~
    Q1145=+1                ;EXTRUSION POINTS ~
    Q1146=+0                ;EXTRUSION LENGTH ~
    Q1149=+0                ;EXTRUSION MODAL
```

29.6 Touch Probe Cycles: Calibration

29.6.1 Fundamentals

Overview

The control must be specifically prepared by the machine manufacturer for the use of a touch probe.

HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.

In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe; otherwise the control cannot provide precise measuring results.

Always calibrate a touch probe in the following cases:

- Initial configuration
- Broken stylus
- Stylus replacement
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Change of active tool axis

The control assumes the calibration values for the active probe system directly after the calibration process. The updated tool data are immediately effective. It is not necessary to repeat the tool call.

During calibration, the control finds the effective length of the stylus and the effective radius of the stylus tip. To calibrate the 3D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The control provides calibration cycles for calibrating the length and the radius:

Cycle		Call	Further information
461	**TS CALIBRATION OF TOOL LENGTH** ■ Calibrating the length	**DEF**-active	Page 1681
462	**CALIBRATION OF A TS IN A RING** ■ Measuring the radius using a ring gauge ■ Measuring the center offset using a ring gauge	**DEF**-active	Page 1683
463	**TS CALIBRATION ON STUD** ■ Measuring the radius using a stud or a calibration pin ■ Measuring the center offset using a stud or a calibration pin	**DEF**-active	Page 1686
460	**CALIBRATION OF TS ON A SPHERE** ■ Measuring the radius using a calibration sphere ■ Measuring the center offset using a calibration sphere	**DEF**-active	Page 1689

Calibrating a touch trigger probe

In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe; otherwise the control cannot provide precise measuring results.

Always calibrate a touch probe in the following cases:

- Initial configuration
- Broken stylus
- Stylus replacement
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Change of active tool axis

During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The control provides calibration cycles for calibrating the length and the radius.

- The control assumes the calibration values for the active probe system directly after the calibration process. The updated tool data are immediately effective. It is not necessary to repeat the tool call.
- Ensure that the touch probe number in the tool table and the touch-probe number in the touch-probe table are identical.

Further information: "Touch probe table tchprobe.tp", Page 1839

Displaying calibration values

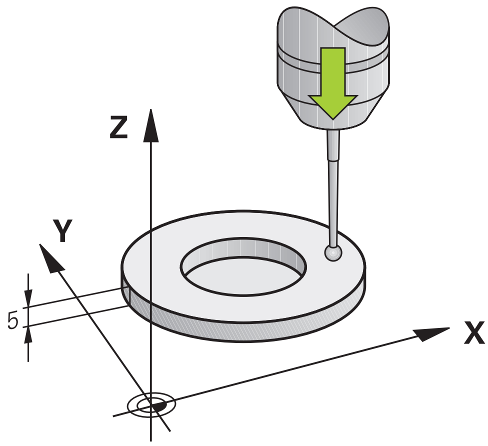
The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL_OF1** (main axis) and **CAL_OF2** (secondary axis).

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

29.6.2 Cycle 461 TS CALIBRATION OF TOOL LENGTH

Application

Refer to your machine manual.



Before starting the calibration cycle, you must set the preset in the spindle axis so that Z=0 on the machine table; you must also pre-position the touch probe above the calibration ring.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

Cycle sequence

1 The control orients the touch probe to the angle **CAL_ANG** specified in the touch probe table (only if your touch probe can be oriented).
2 The control probes from the current position in the negative spindle axis direction at the probing feed rate (column **F** from the touch probe table).
3 The control then retracts the touch probe at rapid traverse (column **FMAX** from the touch probe table) to the starting position.

Notes

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

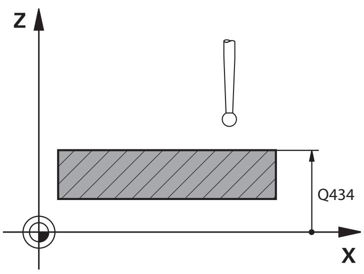
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose, the face of the spindle. The machine manufacturer may also place the tool reference point at a different point.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Cycle parameters

Help graphic	Parameter
	**Q434 Preset for length?** Preset for the length (e.g., height of the calibration ring). The value has an absolute effect. Input: **-99999.9999...+99999.9999**

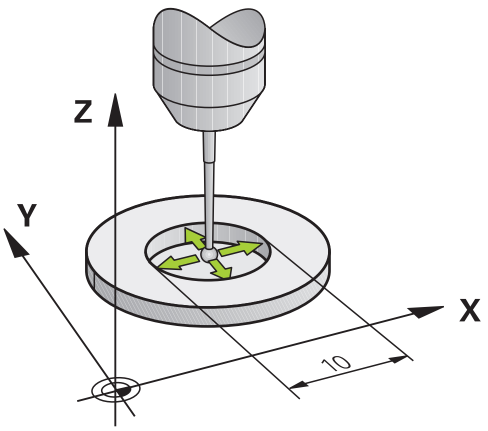
Example

```
11 TCH PROBE 461 TS CALIBRATION OF TOOL LENGTH ~
    Q434=+5                    ;PRESET
```

29.6.3 Cycle 462 CALIBRATION OF A TS IN A RING

Application

Refer to your machine manual.



Before starting the calibration cycle, you need to pre-position the touch probe in the center of the calibration ring and at the required measuring height.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run, the control finds the center point of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. The center offset (**CAL_OF** in touch probe table) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Notes

In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

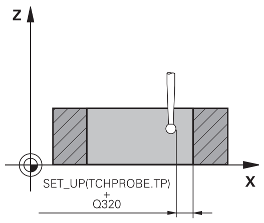
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q407 Radius of ring gauge?** Enter the radius of the ring gauge. Input: **0.0001...99.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q423 Number of probes? Number of measuring points on the diameter. The value has an absolute effect. Input: **3...8**
	Q380 Ref. angle in ref. axis? Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: **0...360**

Example

```
11 TCH PROBE 462 CALIBRATION OF A TS IN A RING ~
    Q407=+5                     ;RING RADIUS ~
    Q320=+0                     ;SET-UP CLEARANCE ~
    Q423=+8                     ;NO. OF PROBE POINTS ~
    Q380=+0                     ;REFERENCE ANGLE
```

29.6.4 Cycle 463 TS CALIBRATION ON STUD

Application

Refer to your machine manual.

Before starting the calibration cycle, you need to pre-position the touch probe above the center of the calibration pin. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration pin.

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column **R** in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. The center offset (CAL_OF in touch probe table) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Note:

In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

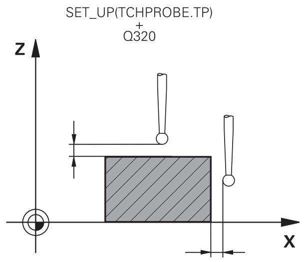
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic	Parameter
	**Q407 Radius of calibr. stud?** Diameter of the ring gauge Input: **0.0001...99.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q423 Number of probes? Number of measuring points on the diameter. The value has an absolute effect. Input: **3...8**
	Q380 Ref. angle in ref. axis? Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: **0...360**

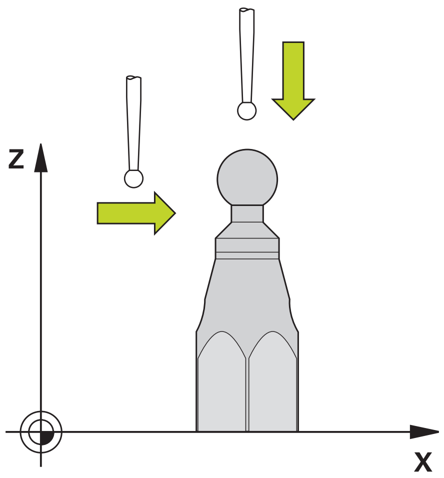
Example

```
11 TCH PROBE 463 TS CALIBRATION ON STUD ~
    Q407=+5                 ;STUD RADIUS ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q301=+1                 ;MOVE TO CLEARANCE ~
    Q423=+8                 ;NO. OF PROBE POINTS ~
    Q380=+0                 ;REFERENCE ANGLE
```

29.6.5 Cycle 460 CALIBRATION OF TS ON A SPHERE (option 17)

Application

Refer to your machine manual.

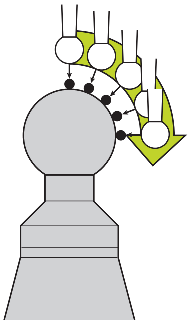


Before starting the calibration cycle, you need to pre-position the touch probe above the center of the calibration sphere. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere.

With Cycle **460** you can calibrate a triggering 3-D touch probe automatically using an exact calibration sphere.

It is also possible to capture 3-D calibration data. Option 92, 3D-ToolComp, is required for this purpose. 3-D calibration data describe the deflection behavior of the touch probe in any probing direction. The 3-D calibration data are stored under TNC:\system\3D-ToolComp*. The **DR2TABLE** column of the tool table references the 3DTC table. The 3-D calibration data are then taken into account when probing. This 3-D calibration is necessary if you want to achieve a very high accuracy with Cycle **444** 3-D Probing (see "Cycle 444 PROBING IN 3-D ", Page 1668).

Cycle sequence



The setting in parameter **Q433** specifies whether you can perform radius and length calibration, or just radius calibration.

Radius calibration Q433=0

1 Clamp the calibration sphere. Ensure the prevention of collisions
2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center
3 The first movement is in the plane, depending on the reference angle (**Q380**)
4 The control then positions the touch probe in touch-probe axis
5 The probing process starts, and the control begins by searching for the equator of the calibration sphere
6 Once the equator has been determined, the radius calibration begins
7 Finally, the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned

Radius and length calibration Q433=1

1 Clamp the calibration sphere. Ensure the prevention of collisions
2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center
3 The first movement is in the plane, depending on the reference angle (**Q380**)
4 The control then positions the touch probe in touch-probe axis
5 The probing process starts, and the control begins by searching for the equator of the calibration sphere
6 Once the equator has been determined, the radius calibration begins
7 The control then retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned
8 The control determines the length of the touch probe at the north pole of the calibration sphere
9 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned

The setting in parameter **Q455** specifies whether you can perform an additional 3-D calibration

3-D calibration Q455= 1...30

1 Clamp the calibration sphere. Ensure the prevention of collisions
2 After calibration of the radius and length, the control retracts the touch probe in touch-probe axis. Then the control positions the touch probe above the north pole
3 The probing process goes from the north pole to the equator in several steps. Deviations from the nominal value, and therefore the specific deflection behavior, are thus determined
4 You can specify the number of probing points between the north pole and the equator. This number depends on input parameter **Q455**. A value between 1 and 30 can be programmed. If you program **Q455**=0, no 3-D calibration will be performed
5 The deviations determined during the calibration are stored in a 3DTC table
6 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned

In order to calibrate the length, the position of the center point (**Q434**) of the calibration sphere relative to the active datum must be known If this is not the case, then HEIDENHAIN recommends against using Cycle **460** to calibrate the length!

One application example for calibrating the length with Cycle **460** is the comparison of two touch probes

Notes

HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

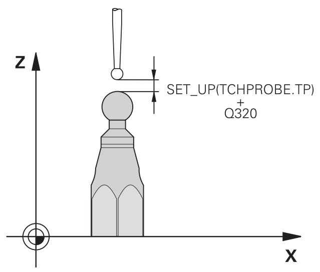
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.
- The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.
- Pre-position the touch probe so that it is located approximately above the center of the calibration sphere.
- Depending on the accuracy of the pre-positioning, finding the equator of the calibration sphere will require a different number of touch points.
- If you program **Q455**=0, the control will not perform a 3-D calibration.
- If you program **Q455**=1 to 30, the control will perform a 3-D calibration of the touch probe. Deviations of the deflection behavior will thus be determined under various angles. If you use Cycle **444**, you should first perform a 3-D calibration.
- If you program **Q455**=1 to 30, a table will be stored under TNC:\system\3D-ToolComp*.
- If there is already a reference to a calibration table (entry in DR2TABLE), this table will be overwritten.
- If there is no reference to a calibration table (entry in DR2TABLE), then, in dependence of the tool number, a reference and the associated table will be created.

Note on programming

- Before a cycle definition you must program a tool call to define the touch-probe axis.

Cycle parameters

Cycle parameters

Help graphic	Parameter
	**Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere being used. Input: **0.0001...99.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: **0**, **1**
	Q423 Number of probes? Number of measuring points on the diameter. The value has an absolute effect. Input: **3...8**
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. The value has an absolute effect. Input: **0...360**
	Q433 Calibrate length (0/1)? Define whether the control will calibrate the touch probe length after radius calibration: **0**: Do not calibrate touch probe length **1**: Calibrate touch probe length Input: **0**, **1**
	Q434 Preset for length? Coordinate of the calibration sphere center. This value must be defined only if length calibration will be carried out. The value has an absolute effect. Input: **-99999.9999...+99999.9999**

Help graphic	Parameter
	Q455 No. of points for 3-D calibrtn.? Enter the number of touch points for 3-D calibration. A value of about 15 touch points is useful. If you enter 0, the control will not perform a 3-D calibration. During 3-D calibration, the deflecting behavior of the touch probe is determined under various angles, and the values are stored in a table. 3D-ToolComp is required for 3-D calibration. Input: **0...30**

Example

```
11 TCH PROBE 460 TS CALIBRATION OF TS ON A SPHERE ~
    Q407=+12.5              ;SPHERE RADIUS ~
    Q320=+0                 ;SET-UP CLEARANCE ~
    Q301=+1                 ;MOVE TO CLEARANCE ~
    Q423=+4                 ;NO. OF PROBE POINTS ~
    Q380=+0                 ;REFERENCE ANGLE ~
    Q433=+0                 ;CALIBRATE LENGTH ~
    Q434=-2.5               ;PRESET ~
    Q455=+15                ;NO. POINTS 3-D CAL.
```

29.7 Touch Probe Cycles: Automatic Kinematics Measurement

29.7.1 Fundamentals (option 48)

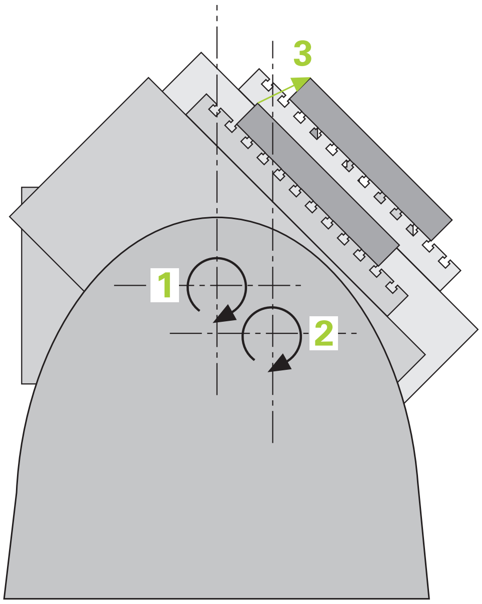
Overview

> The control must be specifically prepared by the machine manufacturer for the use of a touch probe.
> HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.

The control offers the following cycles that enable you to automatically save, restore, check, and optimize the machine kinematics:

Cycle		Call	Further information
450	**SAVE KINEMATICS** (option 48) ■ Storing the active machine kinematic configuration ■ Restoring previously saved kinematic configuration	**DEF-**active	Page 1699
451	**MEASURE KINEMATICS** (option 48) ■ Automatic checking of the machine kinematic configuration ■ Optimizing the machine kinematic configuration	**DEF-**active	Page 1702
452	**PRESET COMPENSATION** (option 48) ■ Automatic checking of the machine kinematic configuration ■ Optimizing the kinematic transformation chain of the machine	**DEF-**active	Page 1717
453	**KINEMATICS GRID** (option 48,option 52) ■ Automatic checking depending on the rotary axis position of the machine kinematic configuration ■ Optimizing the machine kinematic configuration	**DEF-**active	Page 1727

Fundamentals



Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see **1** in the figure) and the kinematic conditions actually existing on the machine (see **2** in the figure). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see **3** in the figure). It is therefore necessary for the model to approach reality as closely as possible.

The **KinematicsOpt** function of the control is an important component that helps you to really meet these complex requirements: a 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are realized as tables or spindle heads. For this purpose, a calibration sphere is attached at any position on the machine table, and measured with a resolution that you define. During cycle definition, you simply define for each rotary axis the area that you want to measure.

From the measured values, the control calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics table.

Requirements

Refer to your machine manual.

Advanced Function Set 1 (option 8) must have been enabled.

Option 48 must have been enabled.

Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

Prerequisites for using KinematicsOpt:

The machine tool builder must have defined the machine parameters for **CfgKinematicsOpt** (no. 204800) in the configuration data.

- **maxModification** (no. 204801) specifies the tolerance limit starting from which the control is to display a message if the changes made to the kinematic data exceed this limit value
- **maxDevCalBall** (no. 204802) defines how much the measured radius of the calibration sphere may deviate from the entered cycle parameter
- **mStrobeRotAxPos** (no. 204803) defines an M function that is specifically configured by the machine tool builder and is used to position the rotary axes

- The 3-D touch probe used for the measurement must be calibrated
- The cycles can only be carried out with the tool axis Z
- A calibration sphere with an exactly known radius and sufficient rigidity must be attached to any position on the machine table
- The kinematics description of the machine must be complete and correct, and the transformation dimensions must have been entered with an accuracy of approx. 1 mm
- The complete machine geometry must have been measured (by the machine tool builder during commissioning)

HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655475-01) or **KKH 80 (ID number 655475-03)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

Notes

HEIDENHAIN only guarantees the proper operation of the probing cycles if HEIDENHAIN touch probes are used.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

A change in the kinematics always changes the preset as well. Basic rotations will automatically be reset to 0. There is a danger of collision!

- After an optimization, reset the preset

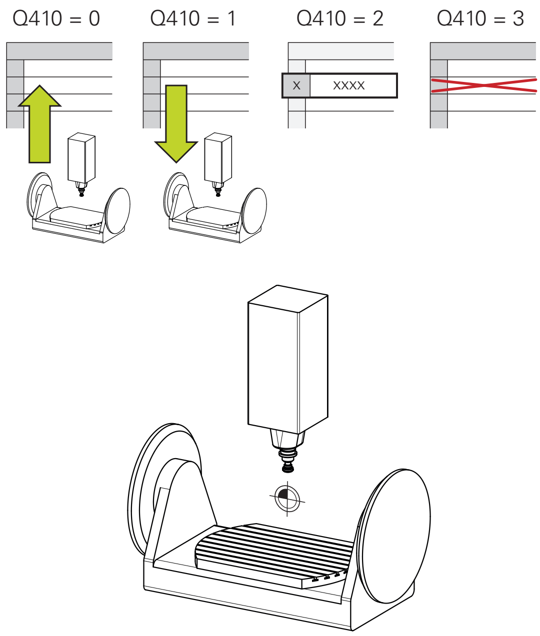
Notes about machine parameters

- In the machine parameter **mStrobeRotAxPos** (no. 204803), the machine manufacturer defines the position of the rotary axes. If an M function has been defined in the machine parameter, you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for **450**).
- If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.

29.7.2 Cycle 450 SAVE KINEMATICS (option 48)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



With touch probe cycle **450** you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.

Notes

Only save and restore data with Cycle **450** while no tool carrier kinematics configuration that includes transformations is active.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Always save the active kinematic model before running a kinematics optimization.
  Advantage:
  - You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g. power failure).
- With the **Restore** mode, note the following:
  - The control can restore saved data only to a matching kinematic configuration
  - A change in the kinematics always changes the preset as well. So redefine the preset, if required.
- The cycle does not restore identical values. It only restores values that differ from the present values. Compensations can only be restored if they had been saved before.

Notes on data management

The control stores the saved data in the file **TNC:\table\DATA450.KD**. This file can be backed up to an external PC with **TNCremo**, for example. If you delete the file, the stored data are removed, too. If the data in the file are changed manually, the data records may become corrupted so that they are unusable.

Operating notes:

- If the file **TNC:\table\DATA450.KD** does not exist, it is generated automatically when Cycle **450** is run.
- Make sure that you delete any empty files with the name **TNC:\table\DATA450.KD** before starting Cycle **450**. If there is an empty memory table (**TNC:\table\DATA450.KD**) without any rows in it, an error message will be issued when running Cycle **450**. In this case, delete the empty memory table and call the cycle again.
- Do not change stored data manually.
- Make a backup of the **TNC:\table\DATA450.KD** file so that you can restore the file, if necessary (e.g. if the data medium is damaged).

Cycle parameters

Help graphic	Parameter
	Q410 Mode (0/1/2/3)? Define whether a kinematic model will be saved or restored: **0**: Save active kinematics **1**: Restore saved kinematics **2**: Display the current memory status **3**: Delete a data record Input: **0**, **1**, **2**, **3**
	Q409/QS409 Name of data record? Number or name of data record identifier. **Q409** has no function if mode 2 has been selected. Wildcards can be used for searches in modes 1 and 3 (Restore and Delete). If the control finds several possible data records because of the wildcards, the control restores the mean values of the data (mode 1) or deletes all selected data records after confirmation (mode 3). You can use the following wildcards in searches: **?**: A single, undefined character **$**: A single alphabetic character (letter) **#**: A single, undefined number *****: An undefined string of any length Input: **0...99999** or max. **255** characters. A total of 16 memory locations is available.

Saving the current kinematics

```
11 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+0                  ;MODE ~
    Q409=+947                ;MEMORY DESIGNATION
```

Restoring data records

```
11 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+1                  ;MODE ~
    Q409=+948                ;MEMORY DESIGNATION
```

Displaying all saved data records

```
11 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+2                  ;MODE ~
    Q409=+949                ;MEMORY DESIGNATION
```

Deleting data records

```
11 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+3                  ;MODE ~
    Q409=+950                ;MEMORY DESIGNATION
```

Log function

After running Cycle **450**, the control creates a log (**tchpr450.txt**) containing the following information:

- Creation date and time of the log
- Name of the NC program from which the cycle was run
- Designator of the current kinematics
- Active tool

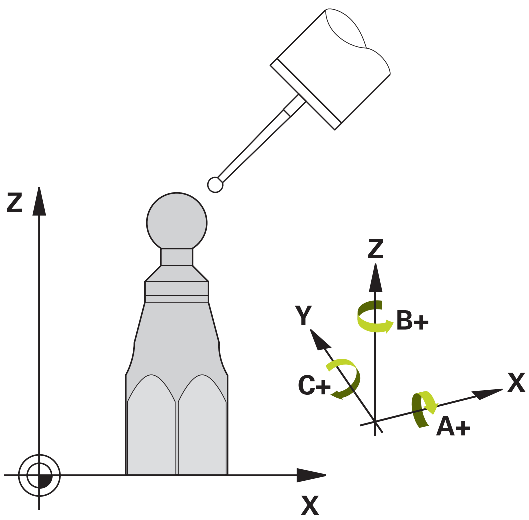
The other data in the log vary depending on the selected mode:

- Mode 0: Logging of all axis entries and transformation entries of the kinematics chain that the control has saved.
- Mode 1: Logging of all transformation entries before and after restoring the kinematics configuration.
- Mode 2: List of the saved data records
- Mode 3: List of the deleted data records

29.7.3 Cycle 451 MEASURE KINEMATICS (option 48)

Application

⚙	Refer to your machine manual. This function must be enabled and adapted by the machine manufacturer.



Touch probe cycle **451** enables you to check and, if required, optimize the kinematics of your machine. Use the 3-D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

The control will determine the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

Cycle sequence

1 Clamp the calibration sphere and check for potential collisions.
2 In **Manual operation** mode, set the preset to the center of the sphere or, if you defined **Q431** = 1 or **Q431** = 3: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3 Select the Program Run operating mode and start the calibration program.
4 The control automatically measures all rotary axes successively in the resolution you defined.

Programming and operating notes:

- If the kinematics data determined in Optimize mode exceed the permissible limit (**maxModification** no. 204801), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.
- During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

The control saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not optimized)
Q145	Optimized standard deviation in the B axis (–1 if axis was not optimized)
Q146	Optimized standard deviation in the C axis (–1 if axis was not optimized)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Positioning direction

The positioning direction of the rotary axis to be measured is determined from the start angle and the end angle that you define in the cycle. A reference measurement is automatically performed at 0°.

Specify the start and end angles in such a way that the same position is not measured twice. A duplicated point measurement (e.g. measuring positions +90° and -270°) is not advisable, but it will not generate an error message.

- Example: Start angle = +90°, end angle = –90°
  - Start angle = +90°
  - End angle = –90°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (-90° - +90°) / (4 – 1) = -60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +30°
  - Measuring point 3 = -30°
  - Measuring point 4 = -90°
- Example: start angle = +90°, end angle = +270°
  - Start angle = +90°
  - End angle = +270°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation = (270° – 90°) / (4 – 1) = +60°
  - Measuring point 1 = +90°
  - Measuring point 2 = +150°
  - Measuring point 3 = +210°
  - Measuring point 4 = +270°

Machines with Hirth-coupled axes

NOTICE

Danger of collision!

In order to be positioned, the axis must move out of the Hirth grid. If necessary, the control rounds the calculated measuring positions so that they fit into the Hirth grid (depending on the start angle, end angle and number of measuring points).

- So remember to leave a large enough safety clearance to prevent any risk of collision between the touch probe and calibration sphere
- Also ensure that there is enough space to reach the safety clearance (software limit switch)

NOTICE

Danger of collision!

Depending on the machine configuration, the control cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine manufacturer, enabling the control to move the rotary axes. The machine manufacturer must have entered the number of the M function in machine parameter **mStrobeRotAxPos** (no. 204803) for this purpose.

- Note the documentation of the machine tool builder

- Define a retraction height greater than 0 if option 2 is not available.
- The measured positions are calculated from the start angle, end angle, and number of measurements for the respective axis and from the Hirth grid.

Example calculation of measuring positions for an A axis:

Start angle **Q411** = –30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Hirth grid = 3°

Calculated stepping angle = (**Q412** – **Q411**) / (**Q414** – 1)

Calculated stepping angle = (90° - (-30°)) / (4 – 1) = 120 / 3 = 40°

Measuring position 1 = **Q411** + 0 * stepping angle = –30° --> –30°

Measuring position 2 = **Q411** + 1 * stepping angle = +10° --> 9°

Measuring position 3 = **Q411** + 2 * stepping angle = +50° --> 51°

Measuring position 4 = **Q411** + 3 * stepping angle = +90° --> 90°

Choice of number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1 or 2), e.g. when commissioning the machine.

You then make a fine optimization with a medium number of measuring points (recommended value = approx. 4). Higher numbers of measuring points do not usually improve the results. Ideally, you should distribute the measuring points evenly over the tilting range of the axis.

This is why you should measure an axis with a tilting range of 0° to 360° at three measuring points, namely at 90°, 180° and 270°. Thus, define a starting angle of 90° and an end angle of 270°.

If you want to check the accuracy accordingly, you can also enter a higher number of measuring points in the **Check** mode.

> If a measuring point has been defined at 0°, it will be ignored because the reference measurement is always done at 0°.

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. The following factors should positively influence the result of measurement:

- On machines with rotary tables/tilting tables: Clamp the calibrating ball as far as possible away from the center of rotation.
- On machines with very large traverse paths: Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

> Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
  - Number of measuring points between 1 and 2
  - Angular step of the rotary axes: Approx. 90°
- **Fine optimization over the entire range of traverse**
  - Number of measuring points between 3 and 6
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
  - Position the calibration sphere in such a way on the machine table that with rotary table axes, there is a large measuring circle or that on swivel head axes, measurement can be made at a representative position (e.g. in the center of the traverse range).
- **Optimization of a specific rotary axis position**
  - Number of measuring points between 2 and 3
  - The measurements are made with the aid of the inclination angle of an axis (**Q413**/**Q417**/**Q421**) around the rotary axis angle at which the workpiece is to be machined later.
  - Position the calibration sphere on the machine table for calibration at the position subsequently intended for machining.
- **Inspecting the machine accuracy**
  - Number of measuring points between 4 and 8
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
- **Determination of the rotary axis backlash**
  - Number of measuring points between 8 and 12
  - The start and end angles should cover the largest possible traverse range of the rotary axes.

Notes on the accuracy

	If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine manual provides further information.

The geometrical and positioning errors of the machine influence the measured values and therefore also the optimization of a rotary axis. For this reason there will always be a certain amount of error.

If there were no geometrical and positioning errors, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning errors are, the greater is the dispersion of measured results when you perform measurements at different positions.

The dispersion of results recorded by the control in the measuring log is a measure of the machine's static tilting accuracy. However, the measuring circle radius and the number and position of measuring points have to be included in the evaluation of accuracy. One measuring point alone is not enough to calculate dispersion. For only one point, the result of the calculation is the spatial error of that measuring point.

If several rotary axes are moved simultaneously, their error values are combined. In the worst case they are added together.

	If your machine is equipped with a feedback-controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.

Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
  - Number of measuring points between 1 and 2
  - Angular step of the rotary axes: Approx. 90°
- **Fine optimization over the entire range of traverse**
  - Number of measuring points between 3 and 6
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
  - Position the calibration sphere in such a way on the machine table that with rotary table axes, there is a large measuring circle or that on swivel head axes, measurement can be made at a representative position (e.g. in the center of the traverse range).
- **Optimization of a specific rotary axis position**
  - Number of measuring points between 2 and 3
  - The measurements are made with the aid of the inclination angle of an axis (**Q413**/**Q417**/**Q421**) around the rotary axis angle at which the workpiece is to be machined later.
  - Position the calibration sphere on the machine table for calibration at the position subsequently intended for machining.
- **Inspecting the machine accuracy**
  - Number of measuring points between 4 and 8
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
- **Determination of the rotary axis backlash**
  - Number of measuring points between 8 and 12
  - The start and end angles should cover the largest possible traverse range of the rotary axes.

Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop, for example because the angle measurement is performed with the motor encoder, this can result in significant error during tilting.

With input parameter **Q432**, you can activate backlash measurement. Enter an angle that the control uses as the traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the control will not measure any backlash.

Backlash measurement is not possible if an M function for positioning the rotary axes is set in the optional **mStrobeRotAxPos** machine parameter (no. 204803) or if the axis is a Hirth axis.

Programming and operating notes:

- The control does not perform an automatic backlash compensation.
- If the measuring circle radius is < 1 mm, the control does not calculate the backlash. The larger the measuring circle radius, the more accurately the control can ascertain the rotary axis backlash (Page 1717).

Notes

Angle compensation is only possible with option 52 KinematicsComp.

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3-D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3-D basic rotation again; otherwise, there is a danger of collision.

- Deactivate the basic rotation before running the cycle.
- Set the preset and the basic rotation again after optimization.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
- Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or set input parameter **Q431** to 1 or 3, respectively.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
- The control ignores cycle definition data that applies to inactive axes.
- A correction in the machine datum (**Q406**=3) is only possible if superimposed rotary axes on the spindle head side or table side are measured.
- If you have activated presetting before the calibration (**Q431** = 1/3), then move the touch probe to the set-up clearance (**Q320** + SET_UP) to a position approximately above the center of the calibration sphere before the start of the cycle.
- Programming in inches: The control always records the log data and results of measurement in millimeters.

Notes about machine parameters

- If the optional machine parameter **mStrobeRotAxPos** (no. 204803) is not equal to –1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0°.
- In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.
- For angle optimization, the machine manufacturer must adapt the configuration correspondingly.

Cycle parameters

Help graphic	Parameter
	Q406 Mode (0/1/2/3)? Define whether the control will check or optimize the active kinematics: **0**: Check the active machine kinematics. The control measures the kinematics in the rotary axes you have defined, but it does not make any changes to the active kinematics. The control displays the measurement results in a measuring log. **1**: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **the rotary axes positions** of the active kinematics. **2**: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then optimizes **angle and position errors**. Software option 52, KinematicsComp, is required for compensation of angle errors. **3**: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then automatically compensates the machine datum. It then optimizes **angle and position errors**. Software option 52, KinematicsComp, is required. Input: **0**, **1**, **2**, **3**
	Q407 Radius of calib. sphere? Enter the exact radius of the calibration sphere being used. Input: **0.0001...99.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q408 Retraction height? **0**: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. **> 0**: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**. The value has an absolute effect. Input: **0...99999.9999**

Help graphic	Parameter
	Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. The value has an absolute effect. Input: **0...360**
	Q411 Starting angle in A axis? Starting angle in the A axis at which the first measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q412 End angle in A axis? End angle in the A axis at which the last measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q413 Angle of incidence in A axis? Angle of incidence in the A axis at which the other rotary axes will be measured. Input: **-359.9999...+359.9999**
	Q414 No. of meas. points in A (0...12)? Number of measuring points the control will use to measure the A axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q415 Starting angle in B axis? Starting angle in the B axis at which the first measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q416 End angle in B axis? End angle in the B axis at which the last measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q417 Angle of incidence in B axis? Angle of incidence in the B axis at which the other rotary axes will be measured. Input: **-359.999...+360.000**

Help graphic	Parameter
	Q418 No. of meas. points in B (0...12)? Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q419 Starting angle in C axis? Starting angle in the C axis at which the first measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q420 End angle in C axis? End angle in the C axis at which the last measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q421 Angle of incidence in C axis? Angle of incidence in the C axis at which the other rotary axes will be measured. Input: **-359.9999...+359.9999**
	Q422 No. of meas. points in C (0...12)? Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q423 Number of probes? Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed and more measuring points increase measurement precision. Input: **3...8**
	Q431 Preset (0/1/2/3)? Define whether the control will automatically set the active preset at the center of the sphere: **0**: Do not set the preset automatically at the center of the sphere: Set the preset manually before the start of the cycle **1**: Set the preset automatically at the center of the sphere before measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle **2**: Set the preset automatically at the center of the sphere after measurement (the active preset will be overwritten): Set the preset manually before the start of the cycle **3**: Set the preset at the center of the sphere before and after measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle Input: **0**, **1**, **2**, **3**

Help graphic	Parameter
	Q432 Angular range of backlash comp.? Define the traversing angle the control will use to measure the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash. Input: **-3...+3**

Saving and checking the kinematics

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+0                ;MODE ~
    Q409=+5                ;MEMORY DESIGNATION
13 TCH PROBE 451 MEASURE KINEMATICS ~
    Q406=+0                ;MODE ~
    Q407=+12.5             ;SPHERE RADIUS ~
    Q320=+0                ;SET-UP CLEARANCE ~
    Q408=+0                ;RETR. HEIGHT ~
    Q253=+750              ;F PRE-POSITIONING ~
    Q380=+0                ;REFERENCE ANGLE ~
    Q411=-90               ;START ANGLE A AXIS ~
    Q412=+90               ;ENDWINKEL A-ACHSE ~
    Q413=+0                ;INCID. ANGLE A AXIS ~
    Q414=+0                ;MEAS. POINTS A AXIS ~
    Q415=-90               ;START ANGLE B AXIS ~
    Q416=+90               ;END ANGLE B AXIS ~
    Q417=+0                ;INCID. ANGLE B AXIS ~
    Q418=+2                ;MEAS. POINTS B AXIS ~
    Q419=-90               ;START ANGLE C AXIS ~
    Q420=+90               ;END ANGLE C AXIS ~
    Q421=+0                ;INCID. ANGLE C AXIS ~
    Q422=+2                ;MEAS. POINTS C AXIS ~
    Q423=+4                ;NO. OF PROBE POINTS ~
    Q431=+0                ;PRESET ~
    Q432=+0                ;BACKLASH, ANG. RANGE
```

Various modes (Q406)

Test mode Q406 = 0

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control records the results of a possible position optimization but does not make any adjustments.

"Optimize position of rotary axes" mode Q406 = 1

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- During this, the control tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
- The machine data are adjusted automatically.

Position and Angle Optimization mode Q406 = 2

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- First the control tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- After angle optimization, the control will perform a position optimization. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.

> Depending on the machine kinematics for correctly determining the angles, HEIDENHAIN recommends performing the measurement once with an inclination angle of 0°.

"Optimize machine datum, position, and angle" mode (Q406 = 3)

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control automatically tries to optimize the datum (option 52, KinematicsComp). In order to use a machine datum to compensate for the angular position of a rotary axis, the rotary axis to be compensated must be nearer to the machine base than the measured rotary axis.
- The control then tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- After angle optimization, the control will perform a position optimization. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.

> For correct determination of the angles, HEIDENHAIN recommends performing the measurement once with an inclination angle of 0°.

Position optimization of the rotary axes with preceding, automatic presetting and measurement of the rotary axis backlash

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 451 MEASURE KINEMATICS ~
    Q406=+1               ;MODE ~
    Q407=+12.5            ;SPHERE RADIUS ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q408=+0               ;RETR. HEIGHT ~
    Q253=+750             ;F PRE-POSITIONING ~
    Q380=+0               ;REFERENCE ANGLE ~
    Q411=-90              ;START ANGLE A AXIS ~
    Q412=+90              ;END ANGLE A AXIS ~
    Q413=+0               ;INCID. ANGLE A AXIS ~
    Q414=+0               ;MEAS. POINTS A AXIS ~
    Q415=-90              ;START ANGLE B AXIS ~
    Q416=+90              ;END ANGLE B AXIS ~
    Q417=+0               ;INCID. ANGLE B AXIS ~
    Q418=+4               ;MEAS. POINTS B AXIS ~
    Q419=+90              ;START ANGLE C AXIS ~
    Q420=+270             ;END ANGLE C AXIS ~
    Q421=+0               ;INCID. ANGLE C AXIS ~
    Q422=+3               ;MEAS. POINTS C AXIS ~
    Q423=+3               ;NO. OF PROBE POINTS ~
    Q431=+1               ;PRESET ~
    Q432=+0.5             ;BACKLASH, ANG. RANGE
```

Log function

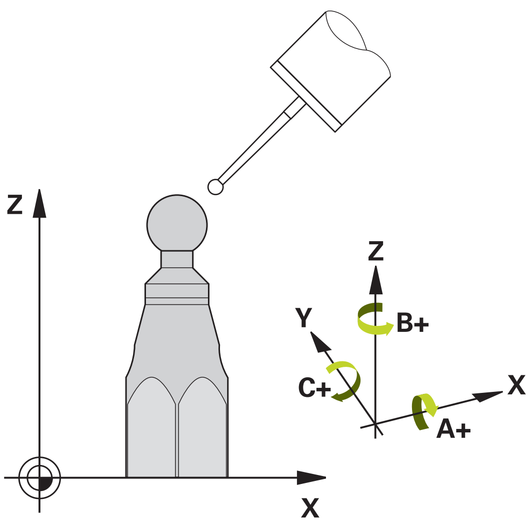
After running Cycle 451, the control will create a log **(TCHPR451.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Check/1=Optimize position/2=Optimize pose)
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Position before optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)

29.7.4 Cycle 452 PRESET COMPENSATION (option 48)

Application

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.



Touch probe cycle **452** optimizes the kinematic transformation chain of your machine (see "Cycle 451 MEASURE KINEMATICS (option 48)", Page 1702). Then the control corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is at the center of the calibration sphere after optimization.

Cycle sequence

> Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

This cycle enables you, for example, to adjust different interchangeable heads so that the workpiece preset applies for all heads.

1. Clamp the calibration sphere
2. Measure the complete reference head with Cycle **451**, and then use Cycle **451** to set the preset in the center of the sphere.
3. Insert the second head
4. Use Cycle **452** to measure the interchangeable head up to the point where the head is changed.
5. Use Cycle **452** to adjust other interchangeable heads to the reference head

If it is possible to leave the calibration sphere clamped to the machine table during machining, you can compensate for machine drift, for example. This procedure is also possible on a machine without rotary axes.

1. Clamp the calibration sphere and check for potential collisions.
2. Set the preset in the calibration sphere.
3. Set the preset on the workpiece, and start machining the workpiece.
4. Use Cycle **452** for preset compensation at regular intervals. The control measures the drift of the axes involved and compensates it in the kinematics description.

Q parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not measured)
Q145	Optimized standard deviation in the B axis (–1 if axis was not measured)
Q146	Optimized standard deviation in the C axis (–1 if axis was not measured)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Notes

In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3-D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3-D basic rotation again; otherwise, there is a danger of collision.

- Deactivate the basic rotation before running the cycle.
- Set the preset and the basic rotation again after optimization.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
- Ensure that all functions for tilting the working plane are reset.
- Before defining the cycle, you must set the preset at the center of the calibration sphere and activate it.
- For rotary axes without separate position encoders, select the measuring points in such a way that you have to traverse an angle of 1° to the limit switch. The control needs this traverse for internal backlash compensation.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while touch probe monitoring is inactive.
- Programming in inches: The control always records the log data and results of measurement in millimeters.

- If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle **450**, so that in case of a failure the most recently active kinematic configuration can be restored.

Notes about machine parameters

- In the machine parameter **maxModification** (no. 204801), the machine manufacturer defines the permissible limit value for modifications of a transformation. If the kinematics data determined exceed the permissible limit value, the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC Start**.
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Cycle parameters

Help graphic	Parameter
	Q407 Radius of calib. sphere? Enter the exact radius of the calibration sphere being used. Input: **0.0001...99.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q408 Retraction height? **0**: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. **> 0**: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**. The value has an absolute effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. The value has an absolute effect. Input: **0...360**
	Q411 Starting angle in A axis? Starting angle in the A axis at which the first measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q412 End angle in A axis? End angle in the A axis at which the last measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q413 Angle of incidence in A axis? Angle of incidence in the A axis at which the other rotary axes will be measured. Input: **-359.9999...+359.9999**

Help graphic	Parameter
	Q414 No. of meas. points in A (0...12)? Number of measuring points the control will use to measure the A axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q415 Starting angle in B axis? Starting angle in the B axis at which the first measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q416 End angle in B axis? End angle in the B axis at which the last measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q417 Angle of incidence in B axis? Angle of incidence in the B axis at which the other rotary axes will be measured. Input: **-359.999...+360.000**
	Q418 No. of meas. points in B (0...12)? Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q419 Starting angle in C axis? Starting angle in the C axis at which the first measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q420 End angle in C axis? End angle in the C axis at which the last measurement will be made. The value has an absolute effect. Input: **-359.9999...+359.9999**
	Q421 Angle of incidence in C axis? Angle of incidence in the C axis at which the other rotary axes will be measured. Input: **-359.9999...+359.9999**
	Q422 No. of meas. points in C (0...12)? Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis. Input: **0...12**
	Q423 Number of probes? Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed and more measuring points increase measurement precision. Input: **3...8**

Help graphic	Parameter
	Q432 Angular range of backlash comp.? Define the traversing angle the control will use to measure the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash. Input: **-3...+3**

Calibration program

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 450 SAVE KINEMATICS ~
    Q410=+0                  ;MODE ~
    Q409=+5                  ;MEMORY DESIGNATION
13 TCH PROBE 452 PRESET COMPENSATION ~
    Q407=+12.5               ;SPHERE RADIUS ~
    Q320=+0                  ;SET-UP CLEARANCE ~
    Q408=+0                  ;RETR. HEIGHT ~
    Q253=+750                ;F PRE-POSITIONING ~
    Q380=+0                  ;REFERENCE ANGLE ~
    Q411=-90                 ;START ANGLE A AXIS ~
    Q412=+90                 ;END ANGLE A AXIS ~
    Q413=+0                  ;INCID. ANGLE A AXIS ~
    Q414=+0                  ;MEAS. POINTS A AXIS ~
    Q415=-90                 ;START ANGLE B AXIS ~
    Q416=+90                 ;END ANGLE B AXIS ~
    Q417=+0                  ;INCID. ANGLE B AXIS ~
    Q418=+2                  ;MEAS. POINTS B AXIS ~
    Q419=-90                 ;START ANGLE C AXIS ~
    Q420=+90                 ;END ANGLE C AXIS ~
    Q421=+0                  ;INCID. ANGLE C AXIS ~
    Q422=+2                  ;MEAS. POINTS C AXIS ~
    Q423=+4                  ;NO. OF PROBE POINTS ~
    Q432=+0                  ;BACKLASH, ANG. RANGE
```

Adjustment of interchangeable heads

The head change function can vary depending on the individual machine tool. Refer to your machine manual.

- Load the second interchangeable head.
- Insert the touch probe
- Measure the interchangeable head with Cycle **452**
- Measure only the axes that have actually been changed (in this example: only the A axis; the C axis is hidden with **Q422**)
- The preset and the position of the calibration sphere must not be changed during the entire process.
- All other interchangeable heads can be adjusted in the same way

Adjusting an interchangeable head

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 452 PRESET COMPENSATION ~
    Q407=+12.5            ;SPHERE RADIUS ~
    Q320=+0               ;SET-UP CLEARANCE ~
    Q408=+0               ;RETR. HEIGHT ~
    Q253=+2000            ;F PRE-POSITIONING ~
    Q380=+45              ;REFERENCE ANGLE ~
    Q411=-90              ;START ANGLE A AXIS ~
    Q412=+90              ;END ANGLE A AXIS ~
    Q413=+45              ;INCID. ANGLE A AXIS ~
    Q414=+4               ;MEAS. POINTS A AXIS ~
    Q415=-90              ;START ANGLE B AXIS ~
    Q416=+90              ;END ANGLE B AXIS ~
    Q417=+0               ;INCID. ANGLE B AXIS ~
    Q418=+2               ;MEAS. POINTS B AXIS ~
    Q419=+90              ;START ANGLE C AXIS ~
    Q420=+270             ;END ANGLE C AXIS ~
    Q421=+0               ;INCID. ANGLE C AXIS ~
    Q422=+0               ;MEAS. POINTS C AXIS ~
    Q423=+4               ;NO. OF PROBE POINTS ~
    Q432=+0               ;BACKLASH, ANG. RANGE
```

The goal of this procedure is to achieve that the workpiece preset remains unchanged after changing rotary axes (head change).

In the following example, the adjustment of a fork head with A and C axes is described. The A axis is changed, whereas the C axis continues being a part of the basic configuration.

- Insert the interchangeable head that will be used as a reference head.
- Clamp the calibration sphere
- Insert the touch probe
- Use Cycle **451** to measure the complete kinematics, including the reference head
- Define the preset (using **Q431** = 2 or 3 in Cycle **451**) after measuring the reference head

Measuring a reference head

```
11 TOOL CALL "TOUCH_PROBE" Z
12 TCH PROBE 451 MEASURE KINEMATICS ~
    Q406=+1              ;MODE ~
    Q407=+12.5           ;SPHERE RADIUS ~
    Q320=+0              ;SET-UP CLEARANCE ~
    Q408=+0              ;RETR. HEIGHT ~
    Q253=+2000           ;F PRE-POSITIONING ~
    Q380=+45             ;REFERENCE ANGLE ~
    Q411=-90             ;START ANGLE A AXIS ~
    Q412=+90             ;END ANGLE A AXIS ~
    Q413=+45             ;INCID. ANGLE A AXIS ~
    Q414=+4              ;MEAS. POINTS A AXIS ~
    Q415=-90             ;START ANGLE B AXIS ~
    Q416=+90             ;END ANGLE B AXIS ~
    Q417=+0              ;INCID. ANGLE B AXIS ~
    Q418=+2              ;MEAS. POINTS B AXIS ~
    Q419=+90             ;START ANGLE C AXIS ~
    Q420=+270            ;END ANGLE C AXIS ~
    Q421=+0              ;INCID. ANGLE C AXIS ~
    Q422=+3              ;MEAS. POINTS C AXIS ~
    Q423=+4              ;NO. OF PROBE POINTS ~
    Q431=+3              ;PRESET ~
    Q432=+0              ;BACKLASH, ANG. RANGE
```

Drift compensation

> **i** This procedure can also be performed on machines without rotary axes.

During machining, various machine components are subject to drift due to varying ambient conditions. If the drift remains sufficiently constant over the range of traverse, and if the calibration sphere can be left on the machine table during machining, the drift can be measured and compensated with Cycle **452**.

- Clamp the calibration sphere
- Insert the touch probe
- Measure the complete kinematics with Cycle **451** before starting the machining process
- Define the preset (using **Q432** = 2 or 3 in Cycle **451**) after measuring the kinematics
- Then set the presets on your workpiece and start the machining process.

Reference measurement for drift compensation

```
11 TOOL CALL "TOUCH_PROBE" Z
12 CYCL DEF 247 PRESETTING ~
    Q339=+1              ;PRESET NUMBER
13 TCH PROBE 451 MEASURE KINEMATICS ~
    Q406=+1              ;MODE ~
    Q407=+12.5           ;SPHERE RADIUS ~
    Q320=+0              ;SET-UP CLEARANCE ~
    Q408=+0              ;RETR. HEIGHT ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q380=+45             ;REFERENCE ANGLE ~
    Q411=+90             ;START ANGLE A AXIS ~
    Q412=+270            ;END ANGLE A AXIS ~
    Q413=+45             ;INCID. ANGLE A AXIS ~
    Q414=+4              ;MEAS. POINTS A AXIS ~
    Q415=-90             ;START ANGLE B AXIS ~
    Q416=+90             ;END ANGLE B AXIS ~
    Q417=+0              ;INCID. ANGLE B AXIS ~
    Q418=+2              ;MEAS. POINTS B AXIS ~
    Q419=+90             ;START ANGLE C AXIS ~
    Q420=+270            ;END ANGLE C AXIS ~
    Q421=+0              ;INCID. ANGLE C AXIS ~
    Q422=+3              ;MEAS. POINTS C AXIS ~
    Q423=+4              ;NO. OF PROBE POINTS ~
    Q431=+3              ;PRESET ~
    Q432=+0              ;BACKLASH, ANG. RANGE
```

- Measure the drift of the axes at regular intervals.
- Insert the touch probe
- Activate the preset in the calibration sphere.
- Use Cycle **452** to measure the kinematics.
- The preset and the position of the calibration sphere must not be changed during the entire process.

Drift compensation

```
11 TOOL CALL "TOUCH_PROBE" Z
13 TCH PROBE 452 PRESET COMPENSATION ~
    Q407=+12.5          ;SPHERE RADIUS ~
    Q320=+0             ;SET-UP CLEARANCE ~
    Q408=+0             ;RETR. HEIGHT ~
    Q253=+9999          ;F PRE-POSITIONING ~
    Q380=+45            ;REFERENCE ANGLE ~
    Q411=-90            ;START ANGLE A AXIS ~
    Q412=+90            ;END ANGLE A AXIS ~
    Q413=+45            ;INCID. ANGLE A AXIS ~
    Q414=+4             ;MEAS. POINTS A AXIS ~
    Q415=-90            ;START ANGLE B AXIS ~
    Q416=+90            ;END ANGLE B AXIS ~
    Q417=+0             ;INCID. ANGLE B AXIS ~
    Q418=+2             ;MEAS. POINTS B AXIS ~
    Q419=+90            ;START ANGLE C AXIS ~
    Q420=+270           ;END ANGLE C AXIS ~
    Q421=+0             ;INCID. ANGLE C AXIS ~
    Q422=+3             ;MEAS. POINTS C AXIS ~
    Q423=+3             ;NO. OF PROBE POINTS ~
    Q432=+0             ;BACKLASH, ANG. RANGE
```

Log function

After running Cycle **452**, the control creates a log (**TCHPR452.html**) containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Measurement uncertainty of rotary axes
  - Position before preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)

Notes on log data

Page 1717

29.7.5 Cycle 453 KINEMATICS GRID (option 48), (option 52)

Application

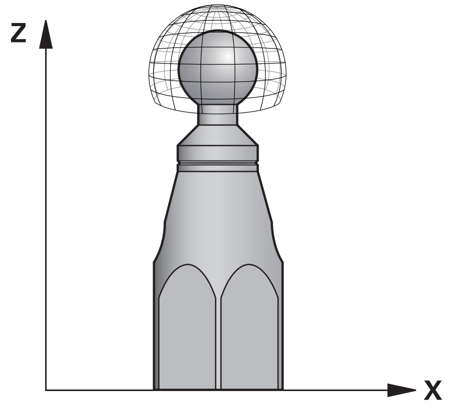
Refer to your machine manual.

KinematicsOpt (software option 48) is required.

KinematicsComp (software option 52) is required.

This function must be enabled and adapted by the machine manufacturer.

To use this cycle, your machine tool builder needs to create and configure a compensation table (*.kco) first and enter some more settings.



Even if your machine was already optimized regarding positioning errors (e.g. via Cycle **451**), residual errors at the Tool Center Point (**TCP**) during tilting of the rotary axes may remain. Such errors occur particularly with swivel-head machines. These can result, for example, from component errors (e.g. a bearing error) with head rotation axes.

Cycle **453 KINEMATICS GRID** enables these errors to be detected and compensated in accordance with the tilting axis positions. Options 48 (**KinematicsOpt**) and 52 (**KinematicsComp**) are required. With this cycle and using a 3-D TS touch probe, you measure a HEIDENHAIN calibration sphere that you have attached to the machine table. The cycle then moves the touch probe automatically to positions in a grid-line arrangement around the calibration sphere. The machine tool builder defines these tilting axis positions. You can arrange the positions in up to three dimensions. (Each dimension is a rotary axis.) After the probing process on the sphere, compensation of the errors can be performed using a multi-dimensional table. The machine tool builder defines this compensation table (*.kco) and specifies its storage location.

When using Cycle **453**, run it at different positions in the workspace. This allows you to check immediately if a compensation with Cycle **453** has the desired positive effect on the machine's accuracy. Only when the desired improvements are achieved with the same compensation values at several positions is such a type of compensation suitable for the respective machine. If this is not the case, then the errors are to be sought outside the rotary axes.

Perform the measurement with Cycle **453** in an optimized condition regarding the rotary axis positioning errors. For this purpose, use e.g. Cycle **451** beforehand.

> HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655475-01) or **KKH 100 (ID number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The control then optimizes the accuracy of your machine. For this purpose, it automatically saves the compensation values resulting from a measurement in a compensation table (*.kco). (This applies to mode **Q406**=1.)

Cycle sequence

1 Clamp the calibration sphere and check for potential collisions.
2 In Manual mode of operation, set the preset to the center of the sphere or, if you defined **Q431=1** or **Q431=3**: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3 Select one of the Program Run operating modes and start the NC program
4 The cycle is executed in accordance with the setting in **Q406** (–1=Delete mode / 0=Test mode / 1=Compensate mode)

> During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

Various modes (Q406)

Delete mode Q406 = -1

- The axes are not moved
- The control writes all values to the compensation table (*.kco), setting them to "0". The result is that no further compensations will be effective for the currently selected kinematics.

Test mode Q406 = 0

- The control probes the calibration sphere.
- The results are saved to a log in html format that is stored in the directory as the current NC program

Compensate mode Q406 = 1

- The control probes the calibration sphere.
- The control writes the deviations to the compensation table (*.kco). The table is updated and the compensation settings are immediately effective.
- The results are saved to a log in html format that is stored in the directory as the current NC program

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. It is recommended to clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

> Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

Notes

> KinematicsOpt (software option 48) is required. KinematicsComp (software option 52) is required.
>
> This function must be enabled and adapted by the machine manufacturer.
>
> Your machine tool builder defines the storage location of the compensation table (*.kco).

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3-D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3-D basic rotation again; otherwise, there is a danger of collision.

- Deactivate the basic rotation before running the cycle.
- Set the preset and the basic rotation again after optimization.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
- Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or you set input parameter **Q431** to 1 or 3, respectively.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
- Programming in inches: The control always records the log data and results of measurement in millimeters.
- If you have activated preset setting before the calibration (**Q431** = 1/3), then move the touch probe by the set-up clearance (**Q320** + **SET_UP**) to a position approximately above the center of the calibration sphere before the start of the cycle.

- If your machine is equipped with a feedback-controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.

Notes about machine parameters

- In the machine parameter **mStrobeRotAxPos** (no. 204803), the machine manufacturer defines the maximum permissible modification of a transformation. If the value is not equal to –1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0°.
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Cycle parameters

Help graphic	Parameter
	Q406 Mode (-1/0/+1) Define whether the control will write a value of 0 to the values of the compensation table (*.kco), will check the currently existing deviations, or will perform a compensation. A log file (*.html) is created. **−1**: Delete values in the compensation table (*.kco). The compensation values for TCP positioning errors are set to 0 in the compensation table (*.kco). The control will not perform any probing. No results will be output to the log (*.html). **0**: Check TCP positioning errors. The control measures the TCP positioning errors based on the rotary axis positions but does not write values to the compensation table (*.kco). The control displays the standard and maximum deviation in a log (*.html). **1**: Compensate TCP positioning errors. The control measures the TCP positioning errors based on the rotary axis positions and writes the deviations to the compensation table (*.kco). The compensations are then immediately effective. The control displays the standard and maximum deviation in a log (*.html). Input: **-1**, **0**, **+1**
	Q407 Radius of calib. sphere? Enter the exact radius of the calibration sphere being used. Input: **0.0001...99.9999**
	Q320 Set-up clearance? Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect. Input: **0...99999.9999** or **PREDEF**
	Q408 Retraction height? **0**: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. **> 0**: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**. The value has an absolute effect. Input: **0...99999.9999**
	Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: **0...99999.9999** or **FMAX**, **FAUTO**, **PREDEF**

Help graphic	Parameter
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. The value has an absolute effect. Input: **0...360**
	Q423 Number of probes? Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed and more measuring points increase measurement precision. Input: **3...8**
	Q431 Preset (0/1/2/3)? Define whether the control will automatically set the active preset at the center of the sphere: **0**: Do not set the preset automatically at the center of the sphere: Set the preset manually before the start of the cycle **1**: Set the preset automatically at the center of the sphere before measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle **2**: Set the preset automatically at the center of the sphere after measurement (the active preset will be overwritten): Set the preset manually before the start of the cycle **3**: Set the preset at the center of the sphere before and after measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle Input: **0**, **1**, **2**, **3**

Probing with Cycle 453

```
11 TCH PROBE 453 KINEMATICS GRID ~
    Q406=+0              ;MODE ~
    Q407=+12.5           ;SPHERE RADIUS ~
    Q320=+0              ;SET-UP CLEARANCE ~
    Q408=+0              ;RETR. HEIGHT ~
    Q253=+750            ;F PRE-POSITIONING ~
    Q380=+0              ;REFERENCE ANGLE ~
    Q423=+4              ;NO. OF PROBE POINTS ~
    Q431=+0              ;PRESET
```

Log function

After running Cycle **453**, the control will create a log **(TCHPR453.html**) and save it in the folder where the current NC program resides. It contains the following data:

- Date and time of protocol creation
- Path of the NC program from which the cycle was run
- Number and name of the currently active tool
- Mode
- Measured data: Standard deviation and maximum deviation
- Information at which position in degrees (°) the maximum deviation occurred
- Number of measuring positions

29.8 Touch Probe Cycles: Automatic Tool Measurement

29.8.1 Fundamentals

Overview

Refer to your machine manual.
Some cycles and functions may not be provided on your machine.
Option 17 is required.
The control must be specifically prepared by the machine manufacturer for the use of a touch probe.
HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

In conjunction with the control's tool measurement cycles, the tool touch probe enables you to measure tools automatically: the compensation values for tool length and radius are stored in the tool table and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Measurement of a stationary tool
- Measurement of a rotating tool
- Measurement of individual teeth

Cycle		Call	Further information
480 **30**	**CALIBRATE TT** ■ Calibrating the tool touch probe	**DEF-** active	Page 1738
481 **31**	**CAL. TOOL LENGTH** ■ Measuring the tool length	**DEF-** active	Page 1740
482 **32**	**CAL. TOOL RADIUS** ■ Measuring the tool radius	**DEF-** active	Page 1743
483 **33**	**MEASURE TOOL** ■ Measuring the tool length and radius	**DEF-** active	Page 1746
484	**CALIBRATE IR TT** ■ Calibrating the tool touch probe (e.g., infrared tool touch probe)	**DEF-** active	Page 1749
485	**MEASURE LATHE TOOL** (option 50) ■ Measurement of turning tools	**DEF-** active	Page 1753

Differences between Cycles 30 to 33 and Cycles 480 to 483

The features and the operating sequences are absolutely identical. There are only the following differences between Cycles **30** to **33** and Cycles **480** to **483**:

- Instead of a selectable parameter for the status of the measurement, Cycles **481** to **483** use the fixed parameter **Q199**.

Setting machine parameters

The touch probe cycles **480**, **481**, **482**, **483**, **484** can be hidden with the optional **hideMeasureTT** machine parameter (no. 128901).

Programming and operating notes:

- Before you start working with the touch probe cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300).
- When measuring a stationary tool, the control will use the feed rate for probing defined in the **probingFeed** machine parameter (no. 122709).

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

n = **maxPeriphSpeedMeas** / (r • 0.0063) where

n:	Spindle speed [rpm]
maxPeriphSpeedMeas:	Maximum permissible cutting speed in m/min
r:	Active tool radius [mm]

The probing feed rate is calculated as follows:
v = measuring tolerance • n with

v:	Probing feed rate [mm/min]
Measuring tolerance	Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
n:	Shaft speed [rpm]

probingFeedCalc (no. 122710) determines the calculation of the probing feed rate:

probingFeedCalc (no. 122710) = **ConstantTolerance**:

The measuring tolerance remains constant—regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The lower you set the maximum permissible rotational speed **maxPeriphSpeedMeas** (no. 122712) and the permissible tolerance **measureTolerance1** (no. 122715), the sooner you will encounter this effect.

probingFeedCalc (no. 122710) = **VariableTolerance**:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The control adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	**measureTolerance1**
30 to 60 mm	2 • **measureTolerance1**
60 to 90 mm	3 • **measureTolerance1**
90 to 120 mm	4 • **measureTolerance1**

probingFeedCalc (No. 122710) = **ConstantFeed**:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

Measuring tolerance = (r • **measureTolerance1**)/ 5 mm) where

r:	Active tool radius [mm]
measureTolerance1:	Maximum permissible error of measurement

Entries in the tool table for milling and turning tools

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
DIRECT.	Cutting direction of the tool for measuring a rotating tool	**Cutting direction (M3 = -)?**
R-OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**Tool offset: radius?**
L-OFFS	Radius measurement: Tool offset between upper edge of stylus and lower edge of tool in addition to **offsetToolAxis**. Default: 0	**Tool offset: length?**
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	No function	0: No offset required because tool tip is to be measured	
End mill	4: four cutting edges	R: Offset required because the tool diameter is greater than the contact plate diameter of the TT	0: No additional offset required during radius measurement. Offset from **offsetToolAxis** (no. 122707) used.
Spherical cutter with a diameter of 10 mm	4: four cutting edges	0: No offset required because the south pole of the ball is to be measured.	5: At a diameter of 10 mm, the tool radius will be defined as offset. If this is not the case, the diameter of the spherical cutter will be measured too far down. So the tool diameter will not be correct.

29.8.2 Cycle 30 or 480 CALIBRATE TT

Application

Refer to your machine manual!

You calibrate the TT with touch probe cycle **30** or **480** (Page 1735). The calibration process runs automatically. The control also measures the center misalignment of the calibration tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

You calibrate the TT with touch probe cycle **30** or **480** .

Touch probe

For the touch probe you use a spherical or cuboid probe contact

Cuboid probe contact

For a cuboid probe contact, the machine manufacturer can store in the optional machine parameters **detectStylusRot** (no. 114315) and **tippingTolerance** (no. 114319) whether the angle of misalignment and tilt angle are determined. Determining the angle of misalignment enables compensation for it when measuring tools. The control displays a warning if the tilt angle is exceeded. The values determined can be seen in the status display of the **TT**.

Further information: "TT tab", Page 161

When clamping the tool touch probe, make sure that the edges of the cuboid probe contact are aligned as parallel to the machine axes as possible. The angle of misalignment should be less than 1° and the tilt angle should be less than 0.3°.

Calibration tool

The calibration tool must be a precisely cylindrical part, for example a cylindrical pin. The resulting calibration values are stored in the control memory and are accounted for during subsequent tool measurement.

Cycle sequence

1 Clamp the calibration tool. The calibration tool must be a precisely cylindrical part, for example a cylindrical pin
2 Manually position the calibration tool in the working plane over the center of the TT
3 Position the calibration tool in the tool axis at approximately 15 mm plus set-up clearance over the TT
4 The first movement of the tool is along the tool axis. The tool is first moved to clearance height, i.e. set-up clearance + 15 mm.
5 The calibration process along the tool axis starts
6 This is followed by calibration in the working plane
7 The control positions the calibration tool in the working plane at a position of TT radius + set-up clearance + 11 mm
8 Then the control moves the tool downwards along the tool axis and the calibration process starts
9 During probing, the control moves in a square pattern
10 The control saves the calibration values and considers them during subsequent tool measurement
11 The control then retracts the stylus along the tool axis to set-up clearance and moves it to the center of the TT

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.

Notes about machine parameters

- Use the machine parameter **CfgTTRoundStylus** (no. 114200) or **CfgT-TRectStylus** (no. 114300) to define the functionality of the calibration cycle. Refer to your machine manual.
  - Use the machine parameter **centerPos** to define the position of the TT within the machine's working space.
- The TT needs to be recalibrated if you change the position of the TT on the table and/or a **centerPos** machine parameter.
- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.

Cycle parameters

Help graphic	Parameter
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the calibration tool above the top of the probe contact (safety zone from **safetyDistToolAx** (no. 114203)). Input: **-99999.9999...+99999.9999**

Example of new format

```
11 TOOL CALL 12 Z
12 TCH PROBE 480 CALIBRATE TT ~
    Q260=+100                    ;CLEARANCE HEIGHT
```

Example of old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 30.0 CALIBRATE TT
13 TCH PROBE 30.1 HEIGHT:+90
```

29.8.3 Cycle 31 or 481 CAL. TOOL LENGTH

Application

Refer to your machine manual!

If you want to measure the tool length, program the touch probe cycle **31** or **482** (Page 1735). Input parameters allow you to select which of the three following methods will be used to measure the tool length:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is stationary.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is stationary.

Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**R-OFFS**).

Cycle for measuring a stationary tool (e.g. for drills)

The control positions the tool to be measured above the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. For this measurement, enter 0 in the tool table under Tool offset: radius (**R-OFFS**).

Cycle for measuring individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). You can enter an additional offset in Tool offset: Length (**L-OFFS**) in the tool table. The control probes the tool radially while it is rotating to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function, set the parameter **PROBING THE TEETH** = 1 in Cycle **31**.

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- Set **stopOnCheck** (no. 122717) to **TRUE**
- You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- You can run an individual tooth measurement for tools with **up to 20 teeth**.
- Cycles **31** and **481** do not support touch probes, turning or dressing tools.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOLGRIND.GRD** table, as well as the wear and compensation data (**LBREAK** and **LTOL**) from the **TOOL.T** table.

Q340: **0** and **1**

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools, see "Tool data for grinding tools (option 156)", Page 263.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured data will be entered in the tool table. **0**: The measured tool length is written to column L of tool table TOOL.T, and the tool compensation is set to DL = 0. If there is already a value in TOOL.T, it will be overwritten. **1**: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation is also available in the Q parameter **Q115**. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T). **2**: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter **Q115**. Nothing is entered under L or DL in the tool table. Input: **0**, **1**, **2** Note the behavior with grinding tools, Page 1741
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input: **-99999.9999...+99999.9999**
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: **0**, **1**

Example of new format

```
11 TOOL CALL 12 Z
12 TCH PROBE 481 CAL. TOOL LENGTH ~
    Q340=+1                 ;CHECK ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q341=+1                 ;PROBING THE TEETH
```

Cycle **31** includes an additional parameter:

Help graphic	Parameter
	Parameter number for result? Parameter number in which the control stores the status of the measurement: **0.0**: Tool is within the tolerance **1.0**: Tool is worn (**LTOL** exceeded) **2.0**: Tool is broken (**LBREAK** exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with **NO ENT** Input: **0...1999**

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 31.0 CAL. TOOL LENGTH
13 TCH PROBE 31.1 CHECK:0
14 TCH PROBE 31.2 HEIGHT::+120
15 TCH PROBE 31.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 31.0 CAL. TOOL LENGTH
13 TCH PROBE 31.1 CHECK:1 Q5
14 TCH PROBE 31.2 HEIGHT:+120
15 TCH PROBE 31.3 PROBING THE TEETH:1
```

29.8.4 Cycle 32 or 482 CAL. TOOL RADIUS

Application

Refer to your machine manual!

If you want to measure the tool radius, program the touch probe cycle **32** or **482** (Page 1735). Input parameters allow you to select which of the two following methods will be used to measure the tool radius:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the face of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). The control probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control will measure the radius of each tooth with the aid of oriented spindle stops.

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- Set **stopOnCheck** (no. 122717) to **TRUE**
- You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- Cycles **32** and **482** do not support touch probes, turning or dressing tools.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOLGRIND.GRD** table, as well as the wear and compensation data (**RBREAK** and **RTOL**) from the **TOOL.T** table.

Q340: **0** and **1**

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools, see "Tool data for grinding tools (option 156)", Page 263.

Notes about machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured data will be entered in the tool table. **0**: The measured tool radius is written to column R of the TOOL.T tool table, and the tool compensation is set to DR = 0. If there is already a value in TOOL.T, it will be overwritten. **1**: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation is also available in the Q parameter **Q116**. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T). **2**: The measured tool radius is compared to the tool radius from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter **Q116**. Nothing is entered under R or DR in the tool table. Input: **0**, **1**, **2**
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input: **-99999.9999...+99999.9999**
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: **0**, **1**

Example of new format

```
11 TOOL CALL 12 Z
12 TCH PROBE 482 CAL. TOOL RADIUS ~
    Q340=+1                 ;CHECK ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q341=+1                 ;PROBING THE TEETH
```

Cycle **32** includes an additional parameter:

Help graphic	Parameter
	Parameter number for result? Parameter number in which the control stores the status of the measurement: **0.0**: Tool is within the tolerance **1.0**: Tool is worn (**RTOL** exceeded) **2.0**: Tool is broken (**RBREAK** exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with **NO ENT** Input: **0...1999**

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 32.0 CAL. TOOL RADIUS
13 TCH PROBE 32.1 CHECK:0
14 TCH PROBE 32.2 HEIGHT:+120
15 TCH PROBE 32.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 32.0 CAL. TOOL RADIUS
13 TCH PROBE 32.1 CHECK:1 Q5
14 TCH PROBE 32.2 HEIGHT:+120
15 TCH PROBE 32.3 PROBING THE TEETH:1
```

29.8.5 Cycle 33 or 483 MEASURE TOOL

Application

Refer to your machine manual!

To measure both the length and radius of a tool, program the touch probe cycle **33** or **483** (Page 1735). This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Input parameters allow you to select which of the two following methods will be used to measure the tool:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

Measuring the tool while it is rotating:

The control measures the tool in a fixed programmed sequence. First, if possible, it measures the tool length, and then the tool radius.

Measuring the individual teeth:

The control measures the tool in a fixed programmed sequence. First it measures the tool radius, then the tool length. The sequence of measurement is the same as for touch probe cycles **31** and **32** as well as **481** and **482**.

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- Set **stopOnCheck** (no. 122717) to **TRUE**
- You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- Cycles **33** and **483** do not support touch probes, turning or dressing tools.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOLGRIND.GRD** table, as well as the wear and compensation data (**LBREAK** , **RBREAK**, **LTOL**, and **RTOL**) from the **TOOL.T** table.

Q340: **0** and **1**

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools, see "Tool data for grinding tools (option 156)", Page 263.

Notes about machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured data will be entered in the tool table. **0**: The measured tool length and the measured tool radius are written to columns L and R of the TOOL.T tool table, and the tool compensation is set to DL = 0 and DR = 0. If there is already a value in TOOL.T, it will be overwritten. **1**: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in the Q parameters **Q115** and **Q116**. If the delta value is greater than the permissible tool length or tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T). **2**: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored values and writes it to the Q parameter **Q115** or **Q116**. Nothing is entered under L, R, or DL, DR in the tool table. Input: **0**, **1**, **2**
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input: **-99999.9999...+99999.9999**
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: **0**, **1**

Example of new format

```
11 TOOL CALL 12 Z
12 TCH PROBE 483 MEASURE TOOL ~
    Q340=+1                 ;CHECK ~
    Q260=+100               ;CLEARANCE HEIGHT ~
    Q341=+1                 ;PROBING THE TEETH
```

Cycle **33** includes an additional parameter:

Help graphic	Parameter
	Parameter number for result? Parameter number in which the control stores the status of the measurement: **0.0**: Tool is within the tolerance **1.0**: Tool is worn (**LTOL** or/and **RTOL** exceeded) **2.0**: Tool is broken (**LBREAK** or/and **RBREAK** exceeded). If you do not wish to use the result of measurement within the NC program, answer the dialog prompt with **NO ENT**. Input: **0...1999**

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 33.0 MEASURE TOOL
13 TCH PROBE 33.1 CHECK:0
14 TCH PROBE 33.2 HEIGHT:+120
15 TCH PROBE 33.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 33.0 MEASURE TOOL
13 TCH PROBE 33.1 CHECK:1 Q5
14 TCH PROBE 33.2 HEIGHT:+120
15 TCH PROBE 33.3 PROBING THE TEETH:1
```

29.8.6 Cycle 484 CALIBRATE IR TT

Application

Cycle **484** allows you to calibrate your tool touch probe (e.g., the wireless infrared TT 460 tool touch probe). You can perform the calibration process with or without manual intervention.

- **With manual intervention**: If you define **Q536** = 0, then the control will stop before the calibration process. You then need to position the calibration tool manually above the center of the tool touch probe.
- **Without manual intervention**: If you define **Q536** = 1, then the control will automatically execute the cycle. You may have to program a prepositioning movement before. This depends on the value of the parameter **Q523 POSITION TT**.

Cycle sequence

> Refer to your machine manual.
> The machine manufacturer defines the functionality of the cycle.

To calibrate the tool touch probe, program the touch probe cycle **484**. In input parameter **Q536**, you can specify whether you want to run the cycle with or without manual intervention.

Touch probe

For the touch probe you use a spherical or cuboid probe contact

Cuboid probe contact:

For a cuboid probe contact, the machine manufacturer can store in the optional machine parameters **detectStylusRot** (no. 114315) and **tippingTolerance** (no. 114319) whether the angle of misalignment and tilt angle are determined. Determining the angle of misalignment enables compensation for it when measuring tools. The control displays a warning if the tilt angle is exceeded. The values determined can be seen in the status display of the **TT**.

Further information: "TT tab", Page 161

> When clamping the tool touch probe, make sure that the edges of the cuboid probe contact are aligned as parallel to the machine axes as possible. The angle of misalignment should be less than 1° and the tilt angle should be less than 0.3°.

Calibration tool:

The calibration tool must be a precisely cylindrical part, for example a cylindrical pin. Enter the exact length and radius of the calibration tool into the TOOL.T tool table. After the calibration, the control stores the calibration values and takes them into account during subsequent tool measurements. The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Q536 = 0: With manual intervention before calibration

Proceed as follows:

- Insert the calibration tool
- Start the calibration cycle
- \> The control interrupts the calibration cycle and displays a dialog.
- Manually position the calibration tool above the center of the tool touch probe.

> Ensure that the calibration tool is located above the measuring surface of the probe contact.

- Press **NC start** to resume cycle sequence
- \> If you have programmed **Q523** = **2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200)

Q536 = 1: Without manual intervention before calibration

Proceed as follows:

- Insert the calibrating tool
- Position the calibration tool above the center of the tool touch probe before the start of the cycle.

> - Ensure that the calibration tool is located above the measuring surface of the probe contact.
> - For a calibration process without manual intervention, you do not need to position the calibration tool above the center of the tool touch probe. The cycle adopts the position from the machine parameters and automatically moves the tool to this position.

- Start the calibration cycle
- > The calibration cycle is executed without stopping.
- > If you have programmed **Q523** = **2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200).

Notes

NOTICE

Danger of collision!

To avoid collisions the tool must be pre-positioned before calling the cycle with **Q536**=1! The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

- Specify whether to stop before cycle start or run the cycle automatically without stopping.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. If you use a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. Major inaccuracies may occur if you use a calibration tool whose diameter is too small and/or that protrudes too far from the chuck.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.
- The TT needs to be recalibrated if you change its position on the table.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.

Cycle parameters

Help graphic	Parameter
	Q536 Stop before running (0=Stop)? Define whether the control will stop before the calibration process or whether the cycle will automatically be executed without a stop: **0**: Stop before the calibration process. The control prompts you to position the calibration tool manually above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press **NC Start** to continue the calibration process or press the the **CANCEL** button to cancel the calibration process. **1**: Without stopping before the calibration process. The control starts the calibration process depending on **Q523**. Before running Cycle **484**, you may have to position the tool above the tool touch probe. Input: **0**, **1**
	Q523 Position of tool probe (0-2)? Position of the tool touch probe: **0**: Current position of the calibration tool. The tool touch probe is below the current position of the calibration tool. If **Q536 = 0**, position the calibration tool manually above the center of the tool touch probe during the cycle. If **Q536 = 1**, you need to position the calibration tool above the center of the tool touch probe before the start of the cycle. **1**: Configured position of the tool touch probe. The control adopts the position from the machine parameter **centerPos** (no. 114201). You do not need to preposition the tool. The calibration tool approaches the position automatically. **2**: Current position of the calibration tool. See **Q523 = 0**. The control additionally writes the determined position (where applicable) to the machine parameter **centerPos** (no. 114201) after calibration. Input: **0**, **1**, **2**

Example

```
11 TOOL CALL 12 Z
12 TCH PROBE 484 CALIBRATE IR TT ~
    Q536=+0                ;STOP BEFORE RUNNING ~
    Q523=+0                ;TT POSITION
```

29.8.7 Cycle 485 MEASURE LATHE TOOL (option 50)

Application

Refer to your machine manual!

Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

Cycle **485 MEASURE LATHE TOOL** is available for the measurement of lathe tools using the tool touch probe from HEIDENHAIN. The control measures the tool in a fixed programmed sequence.

Cycle sequence

1 The control positions the lathe tool to the clearance height
2 The lathe tool is oriented based on the entries in **TO** and **ORI**
3 The control moves the tool to the measuring position in the main axis; traverse movement is interpolated in the main and secondary axes
4 Then the lathe tool moves to the measuring position in the tool axis
5 The tool is measured. Depending on the definition of **Q340**, either tool dimensions are changed or the tool is locked
6 The measuring result is transferred to the result parameter **Q199**
7 After the measurement has been performed, the control positions the tool in the tool axis to the clearance height

Result parameter Q199:

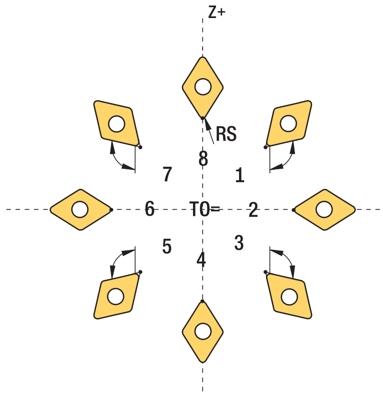
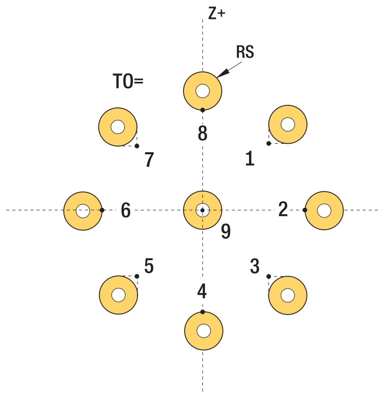
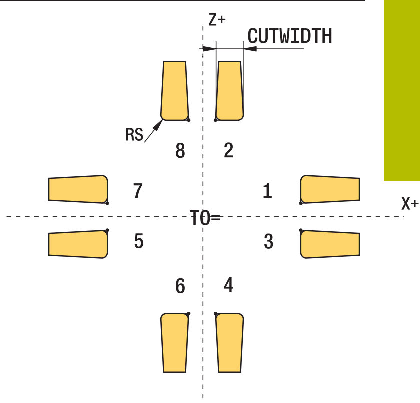
Result	Meaning
0	Tool dimensions within the tolerance **LTOL** / **RTOL** Tool is not locked
1	Tool dimensions outside the tolerance **LTOL** / **RTOL** Tool is locked
2	Tool dimensions outside the tolerance **LBREAK** / **RBREAK** Tool is locked

The cycle uses the following entries from toolturn.trn:

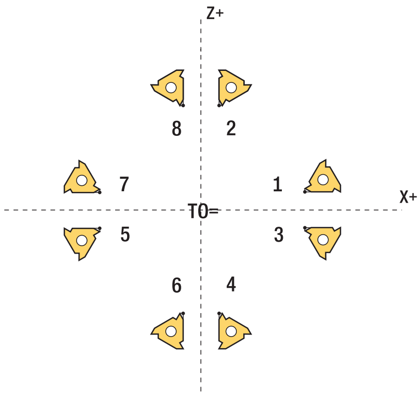
Abbr.	Entries	Dialog
ZL	Tool length 1 (**Z** direction)	**Tool length 1?**
XL	Tool length 2 (**X** direction)	**Tool length 2?**
DZL	Delta value of tool length 1 (**Z** direction), is added to **ZL**	**Oversize in tool length 1?**
DXL	Delta value of tool length 2 (**X** direction), is added to **XL**	**Oversize in tool length 2?**
RS	Cutting edge radius: if contours were programmed with radius compensation **RL** or **RR**, the control takes the cutting edge radius into account in turning cycles, and performs cutting radius compensation	**Cutting edge radius?**
TO	Tool orientation: from the tool orientation, the control determines the position of the tool tip and, depending on the selected tool type, additional information such as the tool angle direction, position of the tool reference point, etc. This information is necessary, for example, for calculating the cutting radius compensation, milling cutter radius compensation, plunge angle, etc.	**Tool orientation?**
ORI	Spindle orientation angle: angle of the indexable insert to the main axis	**Angle of spindle orientation?**
TYPE	Type of turning tool: Roughing tool **ROUGH**, finishing tool **FINISH**, thread tool **THREAD**, recessing tool **RECESS**, button tool **BUTTON**, groove turning tool **RECTURN**	**Type of turning tool**

Page 1755

Tool orientation (TO) that is supported for the following types of turning tools (TYPE)

TYPE	Supported TO with possible limitations	Non-supported TO	
ROUGH, FINISH	■ 1 ■ 7 ■ 2, only **XL** ■ 3, only **XL** ■ 5, only **XL** ■ 6, only **XL** ■ 8, only **ZL**	■ 4 ■ 9	
BUTTON	■ 1 ■ 7 ■ 2, only **XL** ■ 3, only **XL** ■ 5, only **XL** ■ 6, only **XL** ■ 8, only **ZL**	■ 4 ■ 9	
RECESS, RECTURN	■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only **XL** ■ 5, only **XL**	■ 4 ■ 6 ■ 9	

TYPE	Supported TO with possible limitations	Non-supported TO
THREAD	■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only **XL** ■ 5, only **XL**	■ 4 ■ 6 ■ 9



Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- Set **stopOnCheck** (no. 122717) to **TRUE**
- You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

NOTICE

Danger of collision!

If the tool data **ZL** / **DZL** and **XL** / **DXL** deviate by more than ±2 mm from the real tool data, then there is a danger of collision.

- Enter the approximate tool data closer than ±2 mm
- Run the cycle carefully

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before you begin the cycle, you must run a **TOOL CALL** with the tool axis **Z**.
- If you define **YL** and **DYL** with a value outside of ±5 mm, the tool won't reach tool touch probe.
- The cycle does not support **SPB-INSERT** (angular offset). You must enter the value 0 in **SPB-INSERT**, otherwise the control will generate an error message.

Note regarding machine parameters

- The cycle depends on the optional machine parameter **CfgTTRectStylus** (no. 114300). Refer to your machine manual.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Use of the measured values: **0**: The measured values are entered in **ZL** and **XL**. If values are already entered in the tool table, they will be overwritten. **DZL** and **DXL** will be reset to **0**. TL will not be changed **1**: The measured values **ZL** and **XL** are compared with the values from the tool table. These values will not be changed. The control then calculates the deviations of **ZL** and **XL**, and enters these in **DZL** and **DXL**. If the delta values are larger than the permissible wear or breakage tolerance, the control locks the tool (**TL** = Tool Locked). In addition, the deviation is also entered in the Q parameters **Q115** and **Q116** **2**: The measured values **ZL** and **XL** as well as **DZL** and **DXL** are compared with the values from the tool table, but are not changed. If the values are larger than the permissible wear or breakage tolerance, the control locks the tool (**TL** = Tool Locked). Input: **0**, **1**, **2**
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**). Input: **-99999.9999...+99999.9999**

Example

```
11 TOOL CALL 12 Z
12 TCH PROBE 485 MEASURE LATHE TOOL ~
    Q340=+1                  ;CHECK ~
    Q260=+100                ;CLEARANCE HEIGHT
```

30

Application MDI

Application

The **MDI** application allows you to execute individual NC blocks outside of the context of an NC program (e.g., **PLANE RESET**). When you press the **NC Start** key, the control will run the NC blocks separately.

You can also create an NC program step by step. The control memorizes modally effective program information.

Related topics

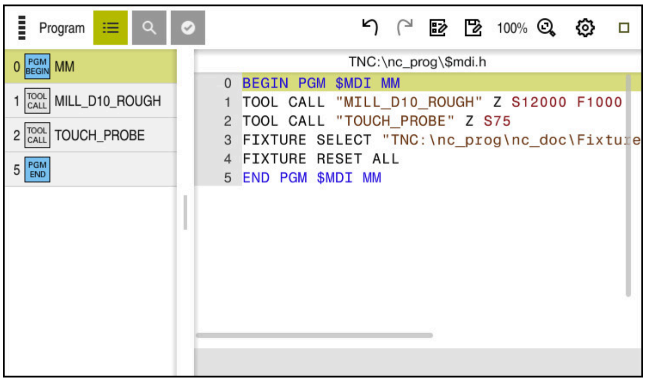
- Creating NC programs

  Further information: "Programming fundamentals", Page 190
- Running NC programs

  Further information: "Program Run", Page 1777

Description of function

If you program using the millimeter unit of measurement, the control will use the NC program **$mdi.h** by default. If you program using the inch unit of measurement, the control will use the NC program **$mdi_inch.h**.



Program workspace in the **MDI** application

The **MDI** application provides the following workspaces:

- **GS** (option 44)

  Further information: "Global Program Settings (GPS, option 44)", Page 1132
- **Help**
- **Positions**

  Further information: "Positions workspace", Page 141
- **Program**

  Further information: "Program workspace", Page 194
- **Simulation**

  Further information: "Simulation Workspace", Page 1405
- **Status**

  Further information: "Status workspace", Page 149
- **Keyboard**

  Further information: "Virtual keyboard of the control bar", Page 1380

Buttons

In the **MDI** application, the function bar provides the following buttons:

Button	Meaning
Klartext programming	If this switch is active, then you are using dialog-guided programming. If this switch is not active, then you are programming in the text editor. **Further information:** "Editing NC programs", Page 202
Insert NC function	The control opens the **Insert NC function** window. **Further information:** "Inserting NC functions", Page 203
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** "Q parameter list window", Page 1271
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** "GOTO function", Page 1383
/ Skip Off/On	Mark NC blocks with a **/** character. NC blocks marked with a **/** character will be ignored during program run as soon as the **Skip /** switch is active. **Further information:** "Hiding NC blocks", Page 1385
Skip /	If this switch is active, the control ignores NC blocks marked with a **/** character. **Further information:** "Hiding NC blocks", Page 1385
; Comment Off/On	Insert or remove a **;** character in front of an NC block. If an NC block begins with a **;** character, then the block is a comment. **Further information:** "Adding comments", Page 1384
FMAX	You activate a feed-rate limitation and define the value. **Further information:** "Feed rate limit F MAX", Page 1781
Editing	The control opens the context menu. **Further information:** "Context menu", Page 1392
Internal stop	If an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. **Further information:** "Interrupting, stopping or canceling program run", Page 1782
Reset program	If you select **Internal stop**, the control activates this button. The control places the cursor back to the beginning of the program and resets any modally effective program information as well as the program run-time.

Modally effective program information

In the **MDI** application, you always run the NC blocks in **Single Block** mode. After the control has run an NC block, the program run is considered to be interrupted.

Further information: "Interrupting, stopping or canceling program run", Page 1782

The block numbers of all NC blocks that you have successively run are shown in green.

The control saves the following data in this state:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

Notes

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying the values of variables by using the window **Q parameter list**
  - Switching the operating modes
- ▶ Restore the contextual reference by repeating the required NC blocks

- In the **MDI** application, you can create and execute NC programs step by step. Then you can use **Save as** to save the current contents with a different file name.
- The following functions are not available in the **MDI** application:
  - Calling an NC program with **PGM CALL**, **SEL PGM**, or **CALL SELECTED PGM**
  - Test Run in the **Simulation** workspace
  - **Manual traverse** and **Approach position** while program run is interrupted
  - **Block scan** function

31

Pallet Machining and Job Lists

31.1 Fundamentals

	Refer to your machine manual. Pallet table management is a machine-dependent function. The standard functional range is described below.

Pallet tables (**.p**) are mainly used in machining centers with pallet changers. The pallet tables call the different pallets (PAL), optionally the fixtures (FIX), and the associated NC programs (PGM). The pallet tables activate all defined presets and datum tables.

Without a pallet changer, you can use pallet tables to successively run NC programs with different presets with just one press of **NC Start**. This type of usage is also called job list.

Tool-oriented machining is possible with pallet tables and with job lists. The control will reduce the number of tool changes, thereby reducing the machining time.

Further information: "Tool-oriented machining", Page 1772

31.1.1 Pallet counter

You can define a pallet counter on the control. This allows you to define a variable number of parts produced, for example during pallet machining with automatic workpiece change.

For this purpose, you define a value in the **TARGET** column of the pallet table. The control repeats the NC programs of this pallet until the nominal value has been reached.

By default, every NC program that has been executed increases the actual value by 1. If, for example, an NC program produces more than one workpiece, you define the value in the **COUNT** column of the pallet table.

Further information: "Pallet table", Page 1864

The control displays the defined nominal value and the current actual value in the **Job list** workspace.

Further information: "Information about the pallet table", Page 1765

31.2 Job list workspace

31.2.1 Fundamentals

Application

In the **Job list** workspace, you edit and execute pallet tables.

Related topics

- Contents of a pallet table

  Further information: "Pallet table", Page 1864
- **Form** workspace for pallets

  Further information: "Form workspace for pallets", Page 1771
- Tool-oriented machining

  Further information: "Tool-oriented machining", Page 1772

Description of function

In the **Job list** workspace, the control displays the individual rows of the pallet table and the status.

Further information: "Information about the pallet table", Page 1765

If you activate the **Edit** switch, the **Insert row** button will be displayed in the action bar and allows you to insert a new table row.

Further information: "Insert row window", Page 1767

When you open a pallet table in **Editor** or **Program Run** operating mode, the control will automatically display the **Job list** workspace. You cannot close this workspace.

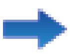
Information about the pallet table

When you open a pallet table, the following information will be displayed in the **Job list** workspace:

Column	Meaning
No column name	Status of the pallet, fixture, or NC program In the **Program Run** operating mode: execution cursor **Further information:** "Status of the pallet, fixture, or NC program", Page 1765
Program	Information about the pallet counter: ■ For rows of the **PAL** type: Current actual value (**COUNT**) and defined nominal value (**TARGET**) of the pallet counter. ■ For rows of the **PGM** type: Value indicating by how much the actual value will be incremented after the execution of the NC program. **Further information:** "Pallet counter", Page 1764 Machining method: ■ Workpiece-oriented machining ■ Tool-oriented machining **Further information:** "Machining method", Page 1766
Sts	Machining status **Further information:** "Machining status", Page 1766

Status of the pallet, fixture, or NC program

The control uses the following icons to display the status:

Icon	Meaning
	The **Pallet**, **Clamping**, or **Program** is locked
	The **Pallet** or **Clamping** is not enabled for machining
	This line is currently being executed in **Program run, single block** or **Program run, full sequence** mode and cannot be edited
	In this line, the program was interrupted manually

Machining method

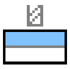
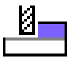
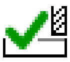
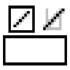
The control uses the following icons to display the machining method:

Icon	Meaning
No icon	Workpiece-oriented machining
 	Tool-oriented machining ▪ Start ▪ End

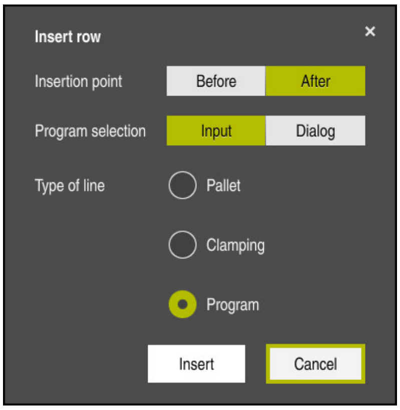
Machining status

The control updates the machining status during program run.

The control uses the following icons to display the machining status:

Icon	Meaning
	Workpiece blank, machining required
	Partially machined, requires further machining
	Completely machined, no further machining required
	Skip machining

Insert row window



Insert row window with **Program** selected

The **Insert row** window provides the following settings:

Setting	Meaning
Insertion point	■ **Before**: Insert a new row before the current cursor position ■ **After**: Insert a new row after the current cursor position
Program selection	■ **Input**: Enter the path of the NC program ■ **Dialog**: Select the NC program via a selection window
Type of line	Corresponds to the **TYPE** column of the pallet table Insert a **Pallet**, **Clamping** or **Program**

You can edit the contents and settings of a row in the **Form** workspace.
Further information: "Form workspace for pallets", Page 1771

Program Run operating mode

You can open the **Program** workspace in addition to the **Job list** workspace. After you have selected a table row with an NC program, the control displays the program contents in the **Program** workspace.

The control uses the execution cursor to indicate which table row is marked for running or is currently being run.

Use the **GOTO Cursor** button to move the execution cursor to the currently selected row of the pallet table.

Further information: "Mid-program startup at any NC block", Page 1768

Mid-program startup at any NC block

To perform a block scan for mid-program startup at an NC block:

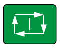
- Open the pallet table in **Program Run** operating mode
- Open the **Program** workspace
- Select the table row with the desired NC program

GOTO Cursor

- Select **GOTO Cursor**
- > The control marks the table row with the execution cursor.
- > The control displays the contents of the NC program in the **Program** workspace.
- Select the desired NC block

Block scan

- Select **Block scan**
- > The control opens the **Block scan** window displaying the values of the NC block.



- Press the **NC Start** key
- > The control starts the block scan.

Notes

- After you have opened a pallet table in **Program Run** operating mode, you can no longer edit this pallet table in **Editor** operating mode.
- In the machine parameter **stopAt** (no. 202101), the machine manufacturer defines when the control will stop program run during the execution of a pallet table.
- In the machine parameter **editTableWhileRun** (no. 202102), the machine manufacturer defines whether you will be allowed to edit the pallet table during program run.
- In the optional machine parameter **resumePallet** (no. 200603), the machine manufacturer defines whether the control will continue program execution after an error message.

31.2.2 Batch Process Manager (option 154)

Application

Batch Process Manager enables you to plan production orders on a machine tool.

The Batch Process Manager software option allows the control to display the following additional information in the **Job list** workspace:

- Times at which manual interventions at the machine are necessary
- Run time of the NC programs
- Availability of the tools
- Whether the NC program is free of errors

Related topics

- **Job list** workspace

  Further information: "Job list workspace", Page 1764
- Editing a pallet table in the **Form** workspace

  Further information: "Form workspace for pallets", Page 1771
- Contents of the pallet table

  Further information: "Pallet table", Page 1864

Requirements

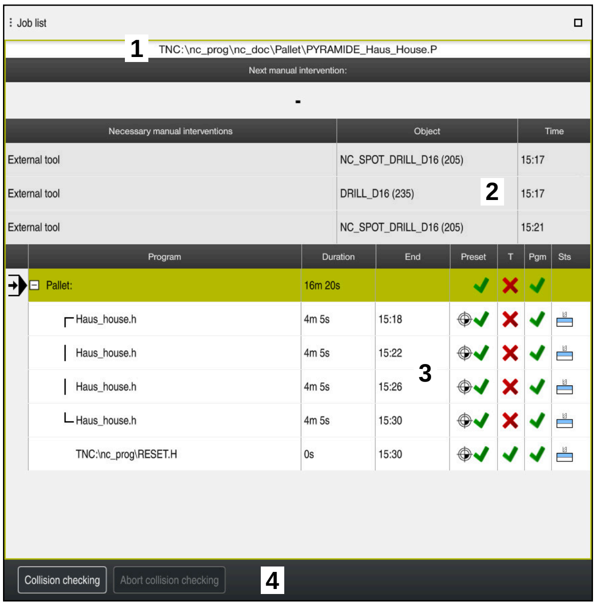
- Software option 22: Pallet Management
- Software option 154: Batch Process Manager

  Batch Process Manager is an expansion to the pallet management feature. Batch Process Manager provides you with all functions available in the **Job list** workspace.
- Tool usage test is active

  The tool usage test function has to be enabled and switched on to ensure you get all information!

  Further information: "Channel settings", Page 1915

Description of function



Job list workspace with **Batch Process Manager** (option 154)

When Batch Process Manager is enabled, the **Job list** workspace provides the following areas:

1. File information bar

   In the file information bar, the control shows the path of the pallet table.
2. Information about necessary manual interventions
   - Time until the next manual intervention
   - Type of intervention
   - Affected object
   - Time of manual intervention
3. Information about and status of the pallet table

   Further information: "Information about the pallet table", Page 1770
4. Action bar

   If the **Edit** switch is active, you can add a new row.

   If the **Edit** switch is inactive, you can use the Dynamic Collision Monitoring (DCM) feature (option 40) to check all NC programs of the pallet table in **Program Run** operating mode.

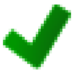
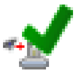
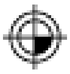
Information about the pallet table

When you open a pallet table, the following information is displayed in the **Job list** workspace:

Column	Meaning
No column name	Status of the pallet, fixture, or NC program In the **Program Run** operating mode: execution cursor **Further information:** "Status of the pallet, fixture, or NC program", Page 1765
Program	Name of the pallet, fixture, or NC program Information about the pallet counter: ■ For rows of the **PAL** type: Current actual value (**COUNT**) and defined nominal value (**TARGET**) of the pallet counter. ■ For rows of the **PGM** type: Value indicating by how much the actual value will be incremented after the execution of the NC program. **Further information:** "Pallet counter", Page 1764 Machining method: ■ Workpiece-oriented machining ■ Tool-oriented machining **Further information:** "Machining method", Page 1766
Duration	Duration of the NC program Only in the **Editor** operating mode
End	Expected point in time when the execution of the NC program will have been completed In the **Editor** operating mode, this column displays a duration, not a point in time
Preset	Status of the workpiece preset: ■ Workpiece preset is defined ■ Check input **Further information:** "Status of the workpiece preset, the tools, and the NC program", Page 1771
T	Status of the tools used: ■ Test completed ■ Test not yet completed ■ Test failed This column is populated only in the **Program Run** operating mode **Further information:** "Status of the workpiece preset, the tools, and the NC program", Page 1771
Pgm	Status of the NC program: ■ Test completed ■ Test not yet completed ■ Test failed **Further information:** "Status of the workpiece preset, the tools, and the NC program", Page 1771
Sts	Machining status **Further information:** "Machining status", Page 1766

Status of the workpiece preset, the tools, and the NC program

The control displays the status using the following icons:

Icon	Meaning
	Test completed
	Test completed Program simulation with active **Dynamic Collision Monitoring (DCM)**, option 40)
	Test failed (e.g., because of expired tool life, danger of collision)
	Test not yet completed
	Incorrect program structure (e.g., pallet does not contain any subprograms)
	Workpiece preset is defined
	Check input You can assign a workpiece preset either to the pallet or to all NC subprograms.

Note

If you edit the job list, the Collision checking completed  status is reset to Check completed .

31.3 Form workspace for pallets

Application

In the **Form** workspace, the control shows the contents of the pallet table for the selected row.

Related topics

- **Job list** workspace
  Further information: "Job list workspace", Page 1764
- Contents of the pallet table
  Further information: "Pallet table", Page 1864
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 1772

Description of function

Form workspace with the contents of a pallet table

A pallet table can have the following types of rows:

- **Pallet**
- **Clamping**
- **Program**

In the **Form** workspace, the control shows the contents of the pallet table. The control shows the contents relevant to the respective type of the selected row.

You can edit the settings in the **Form** workspace or in **Tables** operating mode. The control synchronizes the contents.

By default, the names of the table columns are used to designate the settings options in the form.

The switches provided in the form correspond to the following table columns:

- The **Locked** switch corresponds to the column **LOCK**
- The **Machinable** switch corresponds to the column **LOCATION**

If the control displays an icon next to the input field, a selection window for selecting the contents is available

The **Form** workspace can be selected for pallet tables in **Editor** or **Program Run** operating mode.

31.4 Tool-oriented machining

Application

Tool-oriented machining allows you to machine several workpieces together even on a machine without pallet changer, which reduces tool-change times. You can thus use the pallet management feature even on machines without a pallet changer.

Related topics

- Contents of the pallet table
  Further information: "Pallet table", Page 1864
- Block scan for mid-program startup in a pallet table
  Further information: "Block scan in pallet tables", Page 1792

Requirements

- Software option 22: Pallet Management
- Tool-change macro for tool-oriented machining
- **METHOD** column with the values **TO** or **TCO**
- NC programs with identical tools
  The tools being used must, at least in part, be the same tools.
- **W-STATUS** column with the values **BLANK** or **INCOMPLETE**
- NC programs must not contain the following functions:
  - **FUNCTION TCPM** or **M128** (option 9)
    Further information: "Compensating the tool angle of inclination with FUNCTION TCPM (option 9)", Page 1027
  - **M144** (option 9)
    Further information: "Factoring the tool offset into the calculations with M144 (option 9)", Page 1255
  - **M101**
    Further information: "Automatically inserting a replacement tool with M101", Page 1260
  - **M118**
    Further information: "Activating handwheel superimpositioning with M118", Page 1239
  - Changing the pallet preset
    Further information: "Pallet preset table", Page 1775

Description of function

The following columns of the pallet table are effective for tool-oriented machining:

- **W-STATUS**
- **METHOD**
- **CTID**
- **SP-X** to **SP-W**
  You can enter safety positions for the axes. The control only approaches these positions if the machine tool builder processes them in the NC macros.

Further information: "Pallet table", Page 1864

In the **Job list** workspace, you can activate or deactivate tool-oriented machining for each NC program via the context menu. This will also cause the control to update the **METHOD** column.

Further information: "Context menu", Page 1392

Sequence of tool-oriented machining

1. The entries TO and CTO tell the control that tool-oriented machining is effective for these rows of the pallet table
2. The control executes the NC program with the entry TO up to the TOOL CALL
3. The W-STATUS changes from BLANK to INCOMPLETE and the control enters a value into the CTID field
4. The control executes all other NC programs with the entry CTO up to the TOOL CALL
5. The control uses the next tool for the following machining steps if one of the following situations applies:
   - The next table row contains the entry PAL
   - The next table rowcontains the entry TO or WPO
   - There are rows in the table that do not yet contain the entry ENDED or EMPTY
6. The control updates the entry in the CTID field with each machining operation
7. If all table rows of the group contain the entry ENDED, the control processes the next rows of the pallet table

Block scan for mid-program startup

You can also return to a pallet table after an interruption. The control can show the rows and the NC block at which the interruption occurred.

The control saves the mid-program startup information in the **CTID** column of the pallet table.

The block scan in the pallet table is workpiece-oriented.

After a block scan, the control can resume tool-oriented machining if the tool-oriented machining method TO and CTO is defined in the following rows.

Further information: "Pallet table", Page 1864

The following functions require special attention, particularly for mid-program startup:

- Changing the machine statuses with a miscellaneous function (e.g. M13)
- Writing to the configuration (e.g. WRITE KINEMATICS)
- Traverse range switchover
- Cycle **32**
- Cycle **800**
- Tilting the working plane

Notes

NOTICE

Danger of collision!

Not all pallet tables and NC programs are suitable for tool-oriented machining. With tool-oriented machining, the control no longer executes the NC programs continuously, but divides them at the tool calls. The division of the NC programs allows functions that were not reset to be effective across programs (machine states). This leads to a danger of collision during machining!

- Consider the stated limitations
- Adapt pallet tables and NC programs to the tool-oriented machining
  - Reprogram the program information after each tool in every NC program (e.g. **M3** or **M4**).
  - Reset special functions and miscellaneous functions before each tool in every NC program (e. g., **Tilt the working plane** or **M138**)
- Carefully test the pallet table and associated NC programs in the **Program run, single block** operating mode

- If you want to start machining again, change the W-STATUS to BLANK or remove the previous input.

Notes on mid-program startup

- The entry in the CTID field remains there for two weeks. After this time, mid-programs startup is no longer possible.
- Do not change or delete the entry in the CTID field.
- The data from the CTID field become invalid after a software update.
- The control saves the preset numbers for mid-program startup. If you change this preset, machining is shifted, too.
- Mid-program startup is no longer possible after editing an NC program within tool-oriented machining.

31.5 Pallet preset table

Application

Pallet presets are an easy way to compensate e.g. mechanical differences between individual pallets.

The machine manufacturer defines the pallet preset table.

Related topics

- Contents of the pallet table
  Further information: "Pallet table", Page 1864
- Workpiece preset management
  Further information: "Preset management", Page 949

Description of function

If a pallet preset is active, the workpiece preset is referenced to it.

In the **PALPRES** column of the pallet table, you can enter the corresponding pallet preset for a pallet.

You can also completely align the coordinate system to the pallet by e.g. positioning the pallet preset in the center of a clamping tower.

If a pallet preset is active, the control does not display an icon. You can check the active pallet preset and the defined values in the **Setup** application.

Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1425

Note

NOTICE

Danger of collision!

Despite a basic rotation based on the active pallet preset, the control does not display an icon in the status display. There is a risk of collision during all subsequent axis movements!

- Check the traverse movements of the machine
- Use pallet presets only in conjunction with pallets

If the pallet preset changes, you need to reset the workpiece preset.

Further information: "Setting a preset manually", Page 952

32

Program Run

32.1 Program Run operating mode

32.1.1 Fundamentals

Application

In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either one block at a time or in full sequence.

You also execute pallet tables in this operating mode.

Related topics

- Executing individual NC blocks in the **MDI** application
  Further information: "Application MDI", Page 1759
- Creating NC programs
  Further information: "Programming fundamentals", Page 190
- Pallet tables
  Further information: "Pallet Machining and Job Lists", Page 1763

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

- Copy the NC program and all called files to the **TNC:** drive

Description of function

The following information also applies to pallet tables and job lists.

When you select a new NC program or when an NC program has been completely executed, the cursor is at the beginning of the program.

If you want to start machining at a different NC block, you first need to select the desired NC block by using the **Block scan** function.

Further information: "Block scan for mid-program startup", Page 1786

By default, the control runs NC programs in Full Sequence mode after the **NC Start** key has been pressed. In this mode, the control runs an NC program continuously up to its end, or up to a manual or programmed interruption.

In the **Single Block** mode, you execute each NC block separately by pressing the **NC Start** key.

The control shows the status of the machining process with the **Control-in-operation** icon in the status overview.

Further information: "Status overview on the control bar", Page 147

The **Program Run** operating mode provides the following workspaces:

- **GS** (option 44)

  Further information: "Global Program Settings (GPS, option 44)", Page 1132
- **Positions**

  Further information: "Positions workspace", Page 141
- **Program**

  Further information: "Program workspace", Page 194
- **Simulation**

  Further information: "Simulation Workspace", Page 1405
- **Status**

  Further information: "Status workspace", Page 149
- **Process Monitoring**

  Further information: "Process Monitoring workspace (option 168)", Page 1153

When you open a pallet table, the control will display the **Job list** workspace. You cannot edit this workspace.

Further information: "Job list workspace", Page 1764

Icons and buttons

The **Program Run** operating mode contains the following icons and buttons:

Icon or button	Meaning
	**Open file** With **Open file** you can open a file, for example an NC program. If you open a file, the control closes the file that was already open.
	Execution cursor The execution cursor shows which NC block is currently being run or is marked for running.
Single Block	If this switch is active, then you run each NC block separately with the **NC Start** key. If Single Block mode is selected, then the operating mode's icon in the control bar changes.
Q info	The control opens the **Q parameter list** window, where you can see and edit the current values and descriptions of the variables. **Further information:** "Q parameter list window", Page 1271
Compensation tables	The control opens a selection menu with the following tables: ■ **D** ■ **T-CS** ■ **WPL-CS** **Further information:** "Compensation during program run", Page 1795
FMAX	You activate a feed-rate limitation and define the value. **Further information:** "Feed rate limit F MAX", Page 1781
GOTO Cursor	The control marks the currently selected table row for execution. Active only if a pallet table is open (option 22) **Further information:** "Job list workspace", Page 1764
Skip /	If this switch is active, the control ignores NC blocks marked with a **/** character. **Further information:** "Hiding NC blocks", Page 1385
Pause at M1	If this switch is active, the control pauses program run at the next NC block with **M1**. **Further information:** "Overview of miscellaneous functions", Page 1225
GOTO block number	Mark an NC block to be run without considering any previous NC blocks **Further information:** "GOTO function", Page 1383
Manual traverse	During a program run interruption you can move the axes manually. If **Manual traverse** is active, then the operating mode's icon in the control bar changes. **Further information:** "Manual traverse during an interruption", Page 1785
Edit	If this switch is active, then you can edit the pallet table. Active only if a pallet table is open **Further information:** "Job list workspace", Page 1764
3D ROT	During a program run interruption you can move the axes manually in the tilted working plane (option 8). **Further information:** "Manual traverse during an interruption", Page 1785

Icon or button	Meaning
Approach position	Return to contour after manual traverse of the machine axes during an interruption **Further information:** "Returning to the contour", Page 1793
Block scan	The **Block scan** function allows you to start program run at any desired NC block. The control takes the preceding parts of the NC program up to this NC block into account mathematically; for example, whether the spindle was switched on with **M3**. **Further information:** "Block scan for mid-program startup", Page 1786
Open in the editor	The control opens the NC program in the **Editor** operating mode. Active only if an NC program is open **Further information:** "Programming fundamentals", Page 190
Internal stop	If an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run.
Reset program	If you select **Internal stop**, the control activates this button. The control places the cursor back to the beginning of the program and resets any modally effective program information as well as the program run-time.

Feed rate limit F MAX

The **F MAX** button allows you to reduce the feed rate for all operating modes. The reduction applies to all rapid traverse and feed rate movements. The value you have entered remains active across power cycles.

The **FMAX** button is available in the **MDI** application and in **Editor** operating mode.

When you select the **FMAX** button in the function bar, the control will open the **Feedrate + FMAX** window.

If a feed rate limit is active, the control highlights the **FMAX** button in color and displays the defined value.

You deactivate the feed rate limit by entering a value of 0 in the **Feedrate + FMAX** window.

Interrupting, stopping or canceling program run

There are several ways to stop a program run:

- Interrupt the program run (e.g., with the miscellaneous function **M0**)
- Stop the program run (e.g., with the **NC stop** key)
- Cancel the program run (e.g., with the **NC Stop** key in combination with the **INTERNAL STOP** button)
- Terminate the program run (e.g., with the miscellaneous functions **M2** or **M30**)

During major errors, the control automatically aborts the program run (e.g., during a cycle call with stationary spindle).

Further information: "Message menu on the information bar", Page 1400

If you run your NC program in **Single Block** mode or in the **MDI** application, the control will switch to the interrupted state after the execution of each NC block.

The control shows the current program run status with the **Control-in-operation** icon.

Further information: "Status overview on the control bar", Page 147

You can execute, e.g., the following functions in an interrupted or canceled state:

- Select operating mode
- Manual traverse of axes
- Check Q parameters and change these if necessary using the **Q INFO** function
- Change setting for the optional programmed interruption with **M1**
- Change setting for the programmed skipping of NC blocks with **/**

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- Do not perform the following interactions:
  - Cursor movement to another NC block
  - The jump command **GOTO** to another NC block
  - Editing an NC block
  - Modifying the values of variables by using the window **Q parameter list**
  - Switching the operating modes
- Restore the contextual reference by repeating the required NC blocks

Programmed interruptions

You can set interruptions directly in the NC program. The control interrupts the program run in the NC block containing one of the following inputs:

- Programmed stop **STOP** (with and without miscellaneous function)
- Programmed stop **M0**
- Conditional stop **M1**

Refer to your machine manual.

The miscellaneous function **M6** may also lead to a suspension of the program run. The machine manufacturer sets the functional scope of the miscellaneous functions.

Resuming program run

After stopping the program with the **NC Stop** key or a programmed interruption, you can resume program run by pressing the **NC Start** key.

After canceling the program run with an **Internal stop**, you must start the program run at the beginning of the NC program or use the **Block scan** function.

After an interruption of the program run within a subprogram or program section repeat, you need to use the **Block scan** function for mid-program startup.

Further information: "Block scan for mid-program startup", Page 1786

Modally effective program information

The control saves the following data during a program interruption:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

The control uses the stored data for returning the tool to the contour (**Approach position** button).

Further information: "Returning to the contour", Page 1793

> The saved data remains active until it is reset (e.g., by selecting a program).

Notes

NOTICE
Danger of collision! Program cancellation, manual intervention, or forgotten resetting of NC functions or transformations can lead to the control performing unexpected or undesirable movements. This can lead to workpiece damage or collision. ▶ Rescind all programmed NC functions and transformations within the NC program ▶ Run a simulation before executing an NC program ▶ Check both the general as well as the additional status display for NC functions and transformations, such as an active basic rotation, before executing an NC program ▶ Carefully verify the NC program in the **Single Block** mode

NOTICE
Danger of collision! With NC software 81762x-16, the TNC7 does not support ISO programming. There is a risk of collision during execution due to the missing support. ▶ Use Klartext NC programs exclusively.

- In the **Program Run** operating mode, the control marks active files with the status **M**, such as a selected NC program or tables. If you open such a file in another operating mode, the controls shows the status on the tab of the application bar.
- When moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.
- You can adjust the feed rate and the spindle speed during program run with the potentiometers.
- If you modify the workpiece reference point during a program run interruption, you must re-select the NC block to resume.

  Further information: "Block scan for mid-program startup", Page 1786
- HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.
- The settings in the **GS** workspace have an effect on program run, such as handwheel superimpositioning (option 44).

  Further information: "Global Program Settings (GPS, option 44)", Page 1132

Definitions

Abbreviation	Definition
GS (global program settings)	Global program settings
ACC (active chatter control)	Active Chatter Control

32.1.2 Manual traverse during an interruption

Application

During a program run interruption you can move the machine axes manually.

Use the **Tilt the working plane (3D ROT)** window to assign the reference system for traversing the axes (option 8).

Related topics

- Manual traverse of machine axes
  Further information: "Moving the machine axes", Page 182
- Manual tilting of the working plane (option 8)
  Further information: "Tilting the working plane (option 8)", Page 978

Description of function

When you select **Manual traverse**, you can move the axes with the axis keys of the control.

Further information: "Using axis keys to move the axes ", Page 182

In the **Tilt the working plane (3D ROT)** window, you can select the following functions:

Icon	Function	Meaning
	M-CS machine	Traversing in the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935
	**W-CS workpiece**	Traversing in the workpiece coordinate system **W-CS** **Further information:** "Workpiece coordinate system W-CS", Page 940
	**WPL-CS working plane**	Traversing in the working plane coordinate system **WPL-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 942
	**T-CS tool**	Traversing in the tool coordinate system **T-CS** **Further information:** "Working plane coordinate system WPL-CS", Page 942

When you select one of the functions, the control will display the associated icon in the **Positions** workspace. The control additionally shows the active coordinate system on the **3D ROT** button.

If **Manual traverse** is active, then the operating mode's icon in the control bar changes.

Notes

NOTICE
Danger of collision! During a program interruption, you can move the axes manually (e.g., in order to retract from a hole when the working plane is tilted). There is a risk of collision if the **3-D ROT** setting is incorrect! ▶ It is better to use the **T-CS** function ▶ Use a low feed rate

- On some machines, you may have to press the **NC Start** key while **Manual traverse** is active in order to enable the axis keys.

  Refer to your machine manual.

32.1.3 Block scan for mid-program startup

Application

The **BLOCK SCAN** function allows you to start an NC program at any desired NC block. The control factors workpiece machining up to this NC block into the calculations. For example, the control will switch on the spindle before the start.

Related topics

- Creating NC programs

  Further information: "Programming fundamentals", Page 190
- Pallet tables and job lists

  Further information: "Pallet Machining and Job Lists", Page 1763

Requirement

- The function must be enabled by your machine manufacturer.

  The **Block scan** function must be enabled and configured by your machine manufacturer.

Description of function

If the NC program was interrupted under the following conditions, the control saves the interruption point:

- **Internal stop** button
- Emergency stop
- Power failure

If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute a block scan directly to the point of interruption. The control displays the message when you switch to **Program Run** operating mode for the first time.

You have the following options for a block scan:

- Block scan in the main program, with repetitions if necessary

  Further information: "Performing a single-level block scan", Page 1789
- Multi-level block scan in subprograms and touch probe cycles

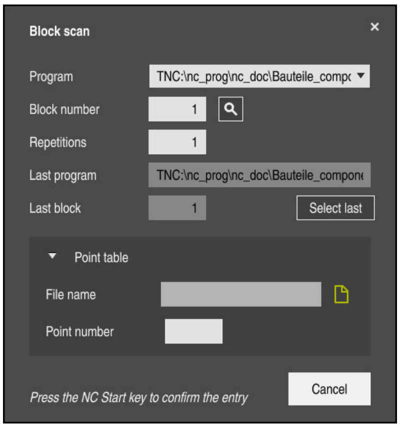
  Further information: "Performing a multi-level block scan", Page 1790
- Block scan in a point table

  Further information: "Block scan in point tables", Page 1791
- Block scan in pallet programs

  Further information: "Block scan in pallet tables", Page 1792

At the start of the block scan, the control resets the data, as with a selection of a new NC program. During a block scan you can activate or deactivate **Single Block** mode.

Block scan window



Block scan window with the interruption point saved and the **Point table** area opened

The **Block scan** window provides the following data:

Row	Meaning
Pallet number	Row number in the pallet table
Program	Path of the active NC program
Block number	Number of the NC block at which program run should start Use the **search** icon to select the NC block in the NC program.
Repetitions	Number of the repetition for mid-program startup if the desired NC block is located within a program-section repeat.
Last pallet number	Pallet number that is active at the time of interruption Select the interruption point by using the **Select last** button.
Last program	Path of the NC program that is active at the time of interruption Select the interruption point by using the **Select last** button.
Last block	Number of the NC block that was active at the time of interruption Select the interruption point by using the **Select last** button.
Point file	Path of the point table In the **Point table** area
Point number	Row in the point table In the **Point table** area

Performing a single-level block scan

To start in an NC program by using a single-level block scan:

- Select the **Program Run** operating mode
- Select **Block scan**
- > The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- Enter the **Program** as needed
- Enter the **Block number**
- Enter the **Repetitions** as needed
- If required, use **Select last** to start at a saved interruption point
- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control will display the **Restore machine status** window.
- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control will display the **Axis sequence for return to contour:** window.
- Press the **NC Start** key
- > Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1794

- Press the **NC Start** key
- > The control resumes execution of the NC program.

Performing a multi-level block scan

If you, for example, start in a subprogram that is called several times by the main program, then use the multi-level block scan. For this, you first go to the desired subprogram call and then continue the block scan. The same procedure is used for called NC programs.

To start in an NC program by using a multi-level block scan:

- Select the **Program Run** operating mode

Block scan

- Select **Block scan**
- > The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- Perform a block scan to the first start-up point:

  Further information: "Performing a single-level block scan", Page 1789

Single Block

- Activate the **Single Block** switch as needed
- Press the **NC Start** key to execute individual NC blocks as needed

Continue block scan

- Select **Continue block scan**
- Define the NC block for mid-program startup
- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control will display the **Restore machine status** window.
- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control will display the **Axis sequence for return to contour:** window.
- Press the **NC Start** key
- > Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1794

Continue block scan

- Select **Continue block scan** again as needed
- Repeat the steps
- Press the **NC Start** key
- > The control resumes execution of the NC program.

Block scan in point tables

To start in a point table:

- Select the **Program Run** operating mode

Block scan

- Select **Block scan**
- > The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- Select **Point table**
- > The control opens the **Point table** area.
- **Point file**: Enter the path of the point table
- **Point number**: Select the row number of the point table for mid-program startup
- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control will display the **Restore machine status** window.
- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control will display the **Axis sequence for return to contour:** window.
- Press the **NC Start** key
- > Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1794

> If you would like to use the block scan function to start in a point pattern, then use the same procedure. Define the desired point for mid-program startup in the **Point number** field. The first point in the point pattern has the number 0.
>
> **Further information:** "Cycles for pattern definition", Page 377

Block scan in pallet tables

To start in a pallet table:

- Select the **Program Run** operating mode

Block scan

- Select **Block scan**
- > The control opens the **Block scan** window.
- **Pallet number**: Enter the row number of the pallet table
- Enter the **Program** as needed
- Enter the **Block number**
- Enter the **Repetitions** as needed

Select last

- If required, use **Select last** to start at a saved interruption point
- Press the **NC Start** key
- > The control starts the block scan and calculates up to the entered NC block.
- > If you have changed the machine status, the control will display the **Restore machine status** window.
- Press the **NC Start** key
- > The control restores the machine status (e.g., **TOOL CALL** or M functions).
- > If you have changed the axis positions, the control will display the **Axis sequence for return to contour:** window.
- Press the **NC Start** key
- > Using the displayed positioning logic, the control moves to the required positions.

> You can also position the axes individually in a self-selected sequence.
>
> **Further information:** "Positioning the axes in a self-selected sequence", Page 1794

> If the program run of a pallet table has been canceled, the control will suggest the most recently selected NC block of the most recently executed NC program as a point of interruption.

Notes

NOTICE

Danger of collision!

If you select an NC block in program run using the **GOTO** function and then execute the NC program, the control ignores all previously programmed NC functions, e.g. transformations. This means that there is a risk of collision during subsequent traversing movements!

- Use **GOTO** only when programming and testing NC programs
- Only use **Block scan** when executing NC programs

NOTICE

Danger of collision!

The **Block scan** function skips over the programmed touch probe cycles. As a result, the result parameters contain no values or, possibly, incorrect values. If the subsequent machining operation uses these result parameters, then there is a risk of collision!

- Use the **Block scan** function at multiple levels

- The control only displays the dialogs required by the process in the pop-up window.
- A **Block scan** always takes place in a workpiece-oriented manner, even if you selected a tool-oriented machining method. After the block scan, the control continues working again in accordance with the selected machining method.

  Further information: "Tool-oriented machining", Page 1772
- Even after an internal stop, the control shows the number of repetitions on the **LBL** tab of the **Status** workspace.

  Further information: "LBL tab", Page 153
- The **Block scan** function must not be used in conjunction with the following functions:
  - Touch probe cycles **0**, **1**, **3**, and **4** during the block scan search phase
- HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.

32.1.4 Returning to the contour

Application

With the **RESTORE POSITION** function, the control moves the tool to the workpiece contour in the following situations:

- Return to the contour after the machine axes were moved during a program interruption that was not performed with the **INTERNAL STOP** function.
- Return to the contour after a block scan (e.g., after an interruption with **INTERNAL STOP**)
- Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption

Related topics

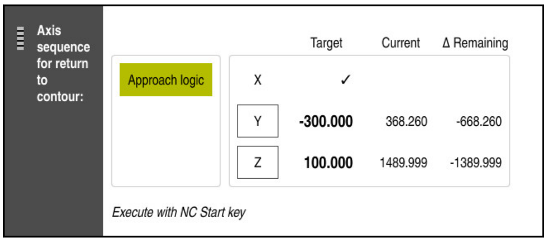
- Manual traverse during program run interruptions
  Further information: "Manual traverse during an interruption", Page 1785
- **Block scan** function
  Further information: "Block scan for mid-program startup", Page 1786

Description of function

If you have selected the **Manual traverse** button, this button will change to **Approach position**.

When you select **Approach position**, the control will open the **Axis sequence for return to contour:** window.

Axis sequence for return to contour: window



Axis sequence for return to contour: window

In the **Axis sequence for return to contour:** window, the control displays all of the axes that are not yet located at the correct position for program execution.

The control suggests a positioning logic for the sequence of the traversing movements. If the tool is located in the tool axis below the position to be approached, then the control offers the tool axis as the first traverse direction. You can also traverse the axes in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 1794

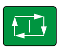
If manual axes are included in the axes to be returned to the contour, then the control will not suggest a positioning logic. As soon as you have correctly positioned the manual axis, the control will suggest a positioning logic for the remaining axes.

Further information: "Positioning manual axes", Page 1795

Positioning the axes in a self-selected sequence

To position the axes in a self-selected sequence:

- Select **Approach position**
- > The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
- Select the desired axis (e.g., **X**)



- Press the **NC Start** key
- > The control moves the axis to the required position.
- > When the axis has reached the correct position, the control shows a checkmark under **Target**.
- Position the remaining axes
- > When all axes have reached their positions, the control closes the window.

Positioning manual axes

To position manual axes:

Approach position

- ▶ Select **Approach position**
- > The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
- ▶ Select the manual axis (e.g., **W**)
- ▶ Position the manual axis to the value shown in the window
- > When a manual axis with encoder has reached the position, the control automatically clears the value.
- ▶ Select **Axis in position**
- > The control saves the position.

Definition

Manual axis
Manual axes are non-driven axes that need to be positioned by the machine operator.

32.2 Compensation during program run

Application

During program run, you can open the selected compensation tables and the active datum table, and edit the values.

Related topics

- Using compensation tables
  Further information: "Tool compensation with compensation tables", Page 1044
- Editing compensation tables in the NC program
  Further information: "Accessing table values ", Page 1809
- Contents and creation of compensation tables
  Further information: "Compensation table *.tco", Page 1868
  Further information: "Compensation table *.wco", Page 1870
- Contents and preparing a datum table
  Further information: "Datum table", Page 957
- Activating a datum table in the NC program
  Further information: "Datum table", Page 1858

Description of function

The control opens the selected tables in **Tables** operating mode.

The changed values do not take effect until the compensation or the datum has been activated again.

32.2.1 Opening tables from within Program Run operating mode

To open the compensation tables from within **Program Run** operating mode:

Compensation tables

- Select **Compensation tables**
- > The control displays a selection menu.
- Select the desired table
  - **D**: Datum table
  - **T-CS**: Compensation table ***.tco**
  - **WPL-CS**: Compensation table ***.wco**
- > The control opens the selected table in **Tables** operating mode.

Notes

NOTICE

Danger of collision!

The control does not consider the changes made to a datum table or compensation table until the values have been saved. You need to activate the datum or compensation value in the NC program again; otherwise, the control will continue using the previous values.

- Make sure to confirm any changes made to the table immediately, e.g., by pressing the **ENT** key
- Activate the datum or compensation value in the NC program again
- Carefully test the NC program after changing the table values

- When you open a table in **Program Run** operating mode, the control will display the status **M** on the tab of the table. This status indicates that this table is active for program run.
- The clipboard allows you to transfer axis positions from the position display to the datum table.

  Further information: "Status overview on the control bar", Page 147

32.3 Retract application

Application

The **Retract** application allows you to disengage the tool from the workpiece after an interruption in power (e.g., retraction of a tap engaged in the workpiece).

You can also retract a tool when the working plane is tilted or retract an inclined tool.

Requirement

- This application must be enabled by your machine manufacturer.

  The machine parameter **retractionMode** (no. 124101) allows the machine manufacturer to define whether the control will display the **Retract** switch during start-up.

Description of function

The **Retract** application provides the following workspaces:

- **Retract**
  Further information: "Retract workspace", Page 1798
- **Positions**
  Further information: "Positions workspace", Page 141
- **Status**
  Further information: "Status workspace", Page 149

The **Retract** application provides the following buttons in the function bar:

Button	Meaning
Retract	Retract the tool with the axis keys or the electronic handwheel
End retraction	Close the **Retract** application The control opens the **End retraction?** window and prompts you to answer a confirmation request.
Start values	Reset the entries in the **A**, **B**, **C**, and **Thread pitch** fields to their original values

You select the **Retract** application by using the **Retract** switch if the following conditions apply during start-up:

- Power interrupted
- No control voltage for the relay
- **Move to ref. point** application

If you have activated a feed rate limit before the power failure occurred, this feed rate limit will still be active. When you select the **Retract** button, the control will display a pop-up window: This window allows you to deactivate the feed rate limit.

Further information: "Feed rate limit F MAX", Page 1781

Retract workspace

The **Retract** workspace provides the following contents:

Row	Meaning
Traversing mode	Traverse mode for retraction: ■ **Machine axes**: Traverse in the machine coordinate system **M-CS** ■ **Tilted system**: Traverse in the working plane coordinate system **WPL-CS** (option 8) ■ **Tool axis**: Traverse in the tool coordinate system **T-CS** (option 8) ■ **Thread**: Traverse in the tool coordinate system **T-CS** with compensating movements of the spindle **Further information:** "Reference systems", Page 934
Kinematics	Name of the active machine kinematics
A, **B**, **C**	Current position of the rotary axes Effective for **Tilted system** traverse mode
Thread pitch	Thread pitch from the **PITCH** column of tool management Effective for **Thread** traverse mode
Direct. of rotation	Direction of rotation of the thread-turning tool: ■ **Right-hand thread** ■ **Left-hand thread** Effective for **Thread** traverse mode
Coordinate system for handwheel superimposition	Coordinate system in which handwheel superimpositioning takes effect Effective for **Tool axis** traverse mode

The control selects the mode of traverse and the associated parameters automatically. If the traverse mode or the parameters have not been correctly preselected, you are able to reset them manually.

Note

NOTICE

Caution: Danger to the tool and workpiece!

A power failure during the machining operation can cause uncontrolled “coasting” or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last saved axis values as the current position, which can deviate from the actual position. Thus, subsequent traverse movements do not correspond to the movements prior to the power failure. If the tool is still in effect during the traverse movements, then the tool and the workpiece can sustain damage through tension!

- ▶ Use a low feed rate
- ▶ Please keep in mind that the traverse range monitoring is not available for non-referenced axes

Example

The power failed while a thread-cutting cycle was being performed in the tilted working plane. You have to retract the tap:

- Switch on the power supply for control and machine
- > The control starts the operating system. This process may take several minutes.
- > The control displays the **Power interrupted** dialog in the **Start/Login** workspace

- Activate the **Retract** switch

- Press **OK**
- > The control compiles the PLC program.

- Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit
- > The control opens the **Retract** application and displays the **Assume position values?** window.
- Compare the displayed position values with the actual position values

- Select **OK**
- > The control closes the **Assume position values?** window
- Select the **Thread** traverse mode as needed
- Enter the thread pitch as needed
- Enter the direction of rotation as needed

- Select **Retract**
- Retract the tool with the axis keys or the handwheel

- Select **End retraction**
- > The control opens the **End retraction?** window and prompts you to answer a confirmation request.

- If the tool was correctly retracted, select **Yes**
- > The control closes the **End retraction?** window and the **Retract** application.

33

Tables

33.1 Tables operating mode

Application

In the **Tables** operating mode you can open various tables and edit them as necessary.

Description of function

If you select **Add**, the control will display the **Quick selection** and **Open File** workspaces.

In the **Quick selection** workspace, you can open some tables directly.

Further information: "Quick selection workspace", Page 1076

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: "Open File workspace", Page 1075

Multiple tables can be open at the same time. The control displays each table in a separate workspace.

If a table is selected for program run or simulation, the control shows the status **M** or **S** on the tab of the application.

You can open the **Table** and **Form** workspaces in every application.

Further information: "Table workspace", Page 1803

Further information: "Form workspace for tables", Page 1807

You can select various functions by using the context menu (e.g., **Copy**).

Further information: "Context menu", Page 1392

Buttons

The **Tables** operating mode provides the following buttons in the function bar:

Button	Meaning
Activate the preset	Activate the selected row of the preset table as a preset. **Further information:** "Preset table", Page 1851
Undo	Undo the last change
Redo	Redo the last undone change
GOTO record	The control opens the **GOTO jump instruction** window. The control jumps to the row number you have defined.
Edit	If the switch is active, you can edit the table.
Insert tool	The control opens the **Insert tool** window that allows you to add a new tool to tool management. **Further information:** "Tool management ", Page 270 When you select the **Append** check box, the control will insert the tool below the last row of the table.
Insert line	The control inserts a row at the end of the table.
Reset row	The control resets all data contained in the row.
Delete tool	The control deletes the tool selected in tool management **Further information:** "Tool management ", Page 270
Delete row	The control deletes the currently selected row.
T INSPECT	The control inspects a tool.
T OUT	The control unloads a tool.
T IN	The controls loads a tool.

33.1.1 Editing the contents of tables

To edit the contents of a table:

- Select the desired table cell

- Activate **Edit**
- > The control enables the values for editing.

> If the **Edit** switch is active, you can edit the contents in both the **Table** workspace and the **Form** workspace.

33.2 Table workspace

Application

In the **Table** workspace, the control shows the contents of a table. The control displays a column with filters and a search function on the left side of some tables.

Description of function

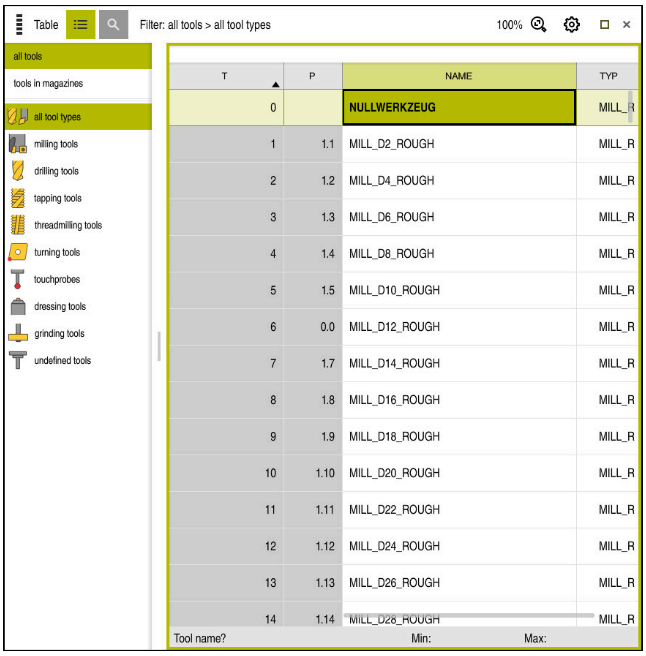


Table workspace

In **Tables** operating mode, the **Table** workspace is open in every application by default.

The control displays the name and path of the file above the header of the table.

When you select the title of a column, the control will sort the table contents by this column.

If the table allows it, you can also edit the table contents in this workspace.

Icons and shortcuts

The **Table** workspace provides the following icons or shortcuts:

Icon or shortcut	Function
	Open the filters **Further information:** "Filters in the Table workspace", Page 1805
	Open the search function **Further information:** "Search column in the Table workspace", Page 1806
100%	Font size of the table When you select the percent value, the control will display icons for increasing and decreasing the font size.
	Set the font size of the table to 100%
	Open the settings in the **Tables** window **Further information:** "Settings in the Table workspace", Page 1806
CTRL+A	Mark all rows
CTRL+BLANK	Mark the active row or end the marking function
SHIFT+↑	Additionally mark the row above
SHIFT+↓	Additionally mark the row below

Filters in the Table workspace

You can filter the tool tables and the **Pocket table**.

Filtering in Tool management

The following filter options are available for tool management:

- **All tools**
- **Magazine tools**

Depending on the filter you have selected (all tools or magazine tools), you can additionally filter by tool type in this area:

- **All types**
- **Milling cutters**
- **Drills**
- **Taps**
- **Thread cutters**
- **Lathe tools**
- **Touch probes**
- **Dressing tools**
- **Grinding tools**
- **Undefined tools**

Filters in the Pocket table

The following filter options are available for the pocket table:

- **All magazines**
- **Main magazine**
- **Spindle**

Depending on the filter you have selected (all magazines, main magazine, or spindle), you can additionally filter by pockets in this area:

- **All pockets**
- **Empty pockets**
- **Assigned pockets**

Search column in the Table workspace

You can search the **Tool management** and **Pocket table** tables.

You can define multiple search conditions in the search function.

Each condition includes the following information:

- Table column, such as **T** or **NAME**
  Use the **Search in** selection menu to select the column.
- Operator, such as **Contains** or **Equal to (=)**
  Use the **Operator** selection menu to select the operator.
- Search term in the **Search for** input field

Settings in the Table workspace

In the **Tables** window, you can influence the contents shown in the **Table** workspace.

The **Tables** window consists of the following areas:

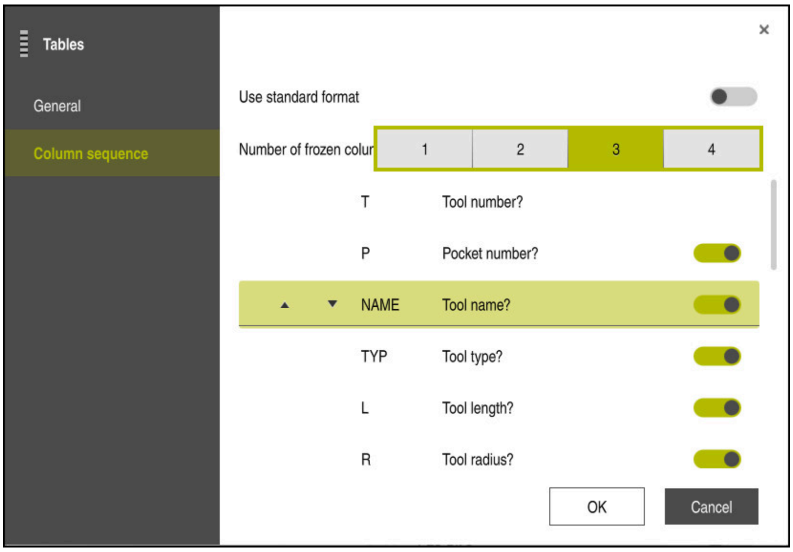
- **General**
- **Column sequence**

General area

The setting selected in the **General** area is modally effective.

If the **Synchronize table and form** switch is active, the cursor will move synchronously. If, for example, you select a different table column in the **Table** workspace, the control will move the cursor synchronously in the **Form** workspace.

Column sequence area



Tables window

In the **Column sequence** area, you define the view for each table.

The **Use standard format** switch allows you to show all columns in the default sequence.

The **Number of frozen columns** switch allows you to define how many columns the control will freeze to the left border of the screen. These columns will remain visible when you navigate further to the right within the table.

The control lists all table columns vertically below each other. Use the switch to select for each column whether it will be shown or hidden.

The control displays a line below the selected number of frozen columns. The control will freeze the columns above this line.

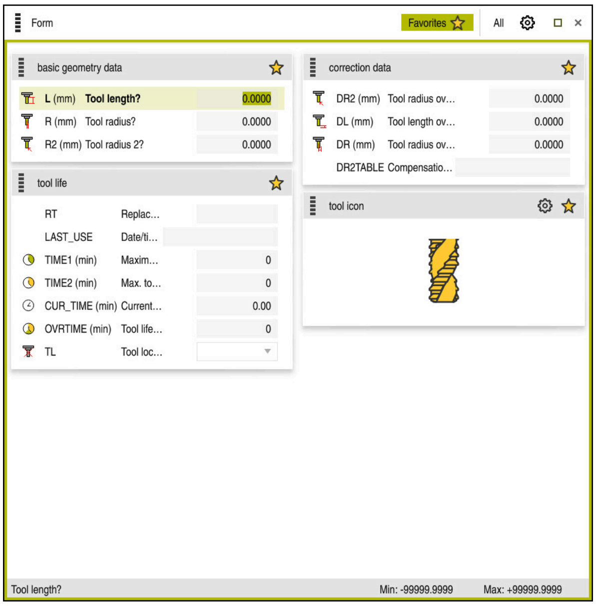
When you select a column, the control will display up and down arrows. Use these arrows to change the sequence of the columns.

33.3 Form workspace for tables

Application

In the **Form** workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.

Description of function



Form workspace in the **Favorites** view

The control displays the following information for each column:

- Icon of the column as needed
- Name of the column
- Unit of measure as needed
- Column description
- Current value

If an input is invalid, the control will display an icon ahead of the input field. When you tap this icon, the control will show the cause of the error (e.g., **Too many characters**).

The control displays the contents of specific tables in groups within the **Form** workspace. In the **All** view, the control shows all groups. Use the **Favorites** function to select individual groups, in order to configure a customized view. Use the handle to arrange the groups.

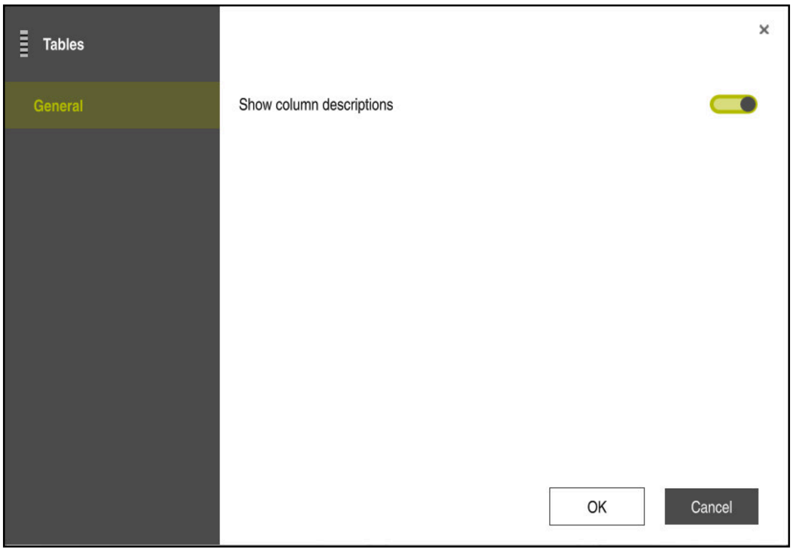
Icons

The **Table** workspace provides the following icons:

Icon or shortcut	Function
	Open the settings in the **Tables** window **Further information:** "Settings in the Form workspace", Page 1809
	Favorite

Settings in the Form workspace

In the **Tables** window, you can select whether the control will show the column descriptions. The selected setting is modally effective.



33.4 Accessing table values

33.4.1 Fundamentals

The **TABDATA** functions allow you to access table values.

These functions enable automated editing of compensation values from within the NC program, for example.

You can access the following tables:

- Tool table ***.t** (read-only access)
- Compensation table ***.tco** (read and write access)
- Compensation table ***.wco** (read and write access)

In each case, the active table is accessed. Read-only access is always possible, whereas write access is possible only during program run. Write access during simulation or during a block scan has no effect.

The control provides the following functions for accessing the table values:

Syntax	Function	Further information
TABDATA READ	Read the value from a table cell	Page 1810
TABDATA WRITE	Write a value to a table cell	Page 1811
TABDATA ADD	Add a value to a table value	Page 1811

If the unit of measure used in the NC program differs from that used in the table, the control will convert the values from **millimeters** to **inches**, and vice versa.

Related topics

- Fundamentals regarding variables
  Further information: "Basics", Page 1268
- Tool table
  Further information: "Tool table tool.t", Page 1814
- Compensation tables
  Further information: "Compensation tables", Page 1868
- Reading values from freely definable tables
  Further information: "Reading a freely definable table with FN 28: TABREAD", Page 1297
- Writing values to freely definably tables
  Further information: "Writing to a freely definable table with FN 27: TABWRITE", Page 1296

33.4.2 Reading table values with TABDATA READ

Application

The function **TABDATA READ** allows you to read a value from a table and save it to a Q parameter.

For example, the **TABDATA READ** function enables you to pre-check the data of the tool to be used to prevent error messages from occurring during program run.

Description of function

Depending on the type of column you want to transfer, you can use **Q**, **QL**, **QR**, or **QS** to save the value. The control will automatically convert the table values to the unit of measure used in the NC program.

Input

11 TABDATA READ Q1 = CORR-TCS COLUMN "DR" KEY "5"	; Save the value in row 5, column **DR**, from the compensation table to **Q1**

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
READ	Reading a table value
Q/QL/QR or **QS**	Type of variable and number in which the control saves the value
TOOL, **CORR-TCS**, or **CORR-WPL**	Read the value from the tool table or a compensation table ***.tco** or ***.wco**
COLUMN	Column name Fixed or variable name
KEY	Row number Fixed or variable name

33.4.3 Writing table values with TABDATA WRITE

Application

The function **TABDATA WRITE** allows you to write a value from a Q parameter into a table

You can use the **TABDATA WRITE** function after a touch probe cycle to enter a necessary tool compensation into the compensation table, for example.

Description of function

Depending on the type of column you want to write to, you can use **Q**, **QL**, **QR**, or **QS** as a transfer parameter.

Input

11 TABDATA WRITE CORR-TCS COLUMN "DR" KEY "3" = Q1	; Write the value from **Q1** to row 5, column **DR**, of the compensation table

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
WRITE	Writing a table value
CORR-TCS or **CORR-WPL**	Writing a value to a compensation table ***.tco** or ***.wco**
COLUMN	Column name Fixed or variable name
KEY	Row number Fixed or variable name
Q/QL/QR or **QS**	Type of variable and number that contains the value to be written

33.4.4 Adding table values with TABDATA ADD

Application

The function **TABDATA ADD** allows you to add a value from a Q parameter to a value contained in the table.

You can use the **TABDATA ADD** function to update a tool compensation value after a measurement has been repeated, for example.

Description of function

Depending on the type of column you want to write to, you can use **Q**, **QL**, or **QR** as a transfer parameter.

In order to write into a compensation table, you need to activate the table.

Further information: "Selecting a compensation table with SEL CORR-TABLE", Page 1046

Input

11 TABDATA ADD CORR-TCS COLUMN "DR" KEY "3" = Q1	; Add the value from **Q1** to row 5, column **DR**, of the compensation table

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
ADD	Adding a value to a table value
CORR-TCS or **CORR-WPL**	Writing a value to a compensation table ***.tco** or ***.wco**
COLUMN	Column name Fixed or variable name
KEY	Row number Fixed or variable name
Q/QL/QR	Type of variable and number that contains the value to be added

33

Tool Tables

33.5.1 Overview

This chapter describes the tool tables of the control.

- Tool table **tool.t**
  Further information: "Tool table tool.t", Page 1814
- Turning tool table **toolturn.trn** (option 50)
  Further information: "Turning tool table toolturn.trn (option 50)", Page 1823
- Grinding tool table **toolgrind.grd** (option 156)
  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 1828
- Dressing tool table **tooldress.drs** (option 156)
  Further information: "Dressing tool table tooldress.drs (option 156)", Page 1836
- Touch probe table **tchprobe.tp**
  Further information: "Touch probe table tchprobe.tp", Page 1839

You can edit the tools, except for the touch probes, in tool management only.

Further information: "Tool management ", Page 270

33.5.2 Tool table tool.t

Application

The tool table **tool.t** contains the data specific to drilling and milling tools. The tool table also contains all tool data that are independent of the technology, such as the tool life **CUR_TIME**.

Related topics

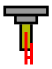
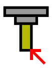
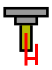
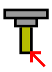
- Editing tool data in tool management
  Further information: "Tool management ", Page 270
- Tool data required for milling or drilling tools
  Further information: "Tool data for milling and drilling tools", Page 259

Description of function

The file name of the tool table is **tool.t** and this table must be stored in the folder **TNC:\table**.

The **tool.t** tool table provides the following parameters:

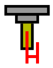
Parameter	Meaning
T	**Tool number?** Row number in the tool table The tool number allows identifying each tool unambiguously, e.g. for calling a tool. **Further information:** "Tool call by TOOL CALL", Page 277 You can define an index after the period. **Further information:** "Indexed tool", Page 250 This parameter applies to all tools, regardless of technology. Input: **0.0...32767.9**

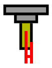
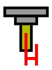
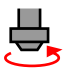
Parameter	Meaning
NAME	**Tool name?** The tool name allows identifying a tool, e.g. for calling a tool. **Further information:** "Tool call by TOOL CALL", Page 277 You can define an index after the period. **Further information:** "Indexed tool", Page 250 This parameter applies to all tools, regardless of technology. Input: **Text width 32**
L 	**Tool length?** Length of tool, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 245 Input: **-99999.9999...+99999.9999**
R 	**Tool radius?** Tool radius, with respect to the tool carrier reference point **Further information:** "Tool carrier reference point", Page 245 Input: **-99999.9999...+99999.9999**
R2 	**Tool radius 2?** Corner radius for the exact definition of the tool for three-dimensional radius compensation, graphic representation and collision monitoring of e.g. ball-nose cutter or toroid cutters. **Further information:** "3D tool compensation (option 9)", Page 1049 Input: **-99999.9999...+99999.9999**
DL 	**Tool length oversize?** Delta value of tool length as a compensation value in connection with touch probe cycles. The control enters corrections automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1449 Is added to the parameter **L** Input: **-999.9999...+999.9999**
DR 	**Tool radius oversize?** Delta value of tool radius as a compensation value in connection with touch probe cycles. The control enters corrections automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1449 Is added to parameter **R** Input: **-999.9999...+999.9999**
DR2 	**Tool radius oversize 2?** Delta value of tool radius 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1449 Is added to parameter **R2** Input: **-999.9999...+999.9999**

Parameter	Meaning
TL	**Tool locked?** Tool is enabled or locked for machining: ■ No value entered: Enabled ■ **L**: Locked The control locks the tool after exceeding maximum tool age **TIME1**, maximum tool age 2 **TIME2** or after exceeding one of the parameters for automatic tool measurement. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: No value, **L**
RT	**Replacement tool?** Number of the replacement tool If the control calls a tool in a TOOL CALL and the tool is not available or locked, the control inserts the replacement tool. If **M101** is active and the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool and inserts the replacement tool at a suitable location. **Further information:** "Automatically inserting a replacement tool with M101", Page 1260 If the replacement tool is not available or locked, the control inserts the replacement tool of the replacement tool. You can define an index after the period. **Further information:** "Indexed tool", Page 250 If you define the value 0, the control will not use a replacement tool. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: **0.0...32767.9**
TIME1	**Maximum tool age?** Maximum tool age in minutes If the current tool age **CUR_TIME** exceeds the **TIME1** value, the control locks the tool and displays an error message when the tool is called the next time. The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99999**

Parameter	Meaning
TIME2	**Max. tool age for TOOL CALL?** Maximum tool age 2 in minutes The control inserts a replacement tool in the cases below: ■ When the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool. The control no longer inserts the tool when the tool is called. If a replacement tool **RT** is defined and available in the magazine, the control inserts the replacement tool. If no replacement tool is available, the control will display an error message. ■ If **M101** is active and the current tool age **CUR_TIME** exceeds the **TIME2** value, the control locks the tool and inserts the replacement tool **RT** at a suitable location. **Further information:** "Automatically inserting a replacement tool with M101", Page 1260 This parameter applies to all tools, regardless of technology. Input: **0...99999**
CUR_TIME	**Current tool age?** The current tool age equals the time during which the tool is cutting a workpiece. The control counts this time automatically and enters the current tool age in minutes. This parameter applies to all tools, regardless of technology. Input: **0...99999.99**
TYPE	**Tool type?** Depending on the selected tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Tool types", Page 255 **Further information:** "Tool management ", Page 270 This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: **MILL**, **MILL_R**, **MILL_F**, **BALL**, **TORUS**, **DRILL**, **TAP**, **CENT**, **TURN**, **TCHP**, **REAM**, **CSINK**, **TSINK BOR**, **BCKBOR**, **GF**, **GSF**, **EP**, **WSP**, **BGF**, **ZBGF**, **GRIND**, and **DRESS**
DOC	**Tool description** This parameter applies to all tools, regardless of technology. Input: **Text width 32**
PLC	**PLC status?** Tool information for the PLC Refer to your machine manual. This parameter applies to all tools, regardless of technology. Entry: **%00000000...%11111111**
LCUTS	**Tooth length in the tool axis?** Length of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring. Input: **-99999.9999...+99999.9999**

Parameter	Meaning
LU	**Usable length of the tool?** Usable length of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of e.g. necks of end mills. Input: **0.0000...999.9999**
RN	**Neck radius of the tool?** Neck radius for the exact definition of the tool for graphic simulation and collision monitoring of e.g. neck of end mills or side milling cutters. Only if the useful length **LU** is longer than the **LCUTS** length of the cutting edge can the tool contain a neck radius **RN**. Input: **0.0000...999.9999**
ANGLE	**Maximum plunge angle?** Maximum plunge angle of the tool for reciprocating plunge-cutting in the cycles. Input: **-360.00...+360.00**
CUT	**Number of teeth?** Number of teeth of the tool for automatic tool measurement or cutting data calculation. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1734 **Further information:** "Cutting data calculator", Page 1397 This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0...99**
TMAT	**Tool material?** Tool material from the tool material table **TMAT.tab** for cutting data calculation. **Further information:** "Table for tool materials TMAT.tab", Page 1861 Selection by means of a selection window Input: **Text width 32**
CUTDATA	**Cutting data table?** **Further information:** "Cutting data calculator", Page 1397 Select cutting data table with the ***.cut** or ***.cutd** file extensions for cutting data calculation. **Further information:** "Cutting data table *.cut", Page 1862 Selection by means of a selection window Entry: **Text width 20**

Parameter	Meaning
LTOL 	**Wear tolerance: length?** Permitted tool length deviation in wear detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1734 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0...9.9999**
RTOL 	**Wear tolerance: radius?** Permitted tool radius deviation in wear detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1734 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0...9.9999**
R2TOL	**Wear tolerance: Radius 2?** Permitted tool radius 2 deviation in wear detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1734 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0...9.9999**
DIRECT 	**Cutting direction?** Cutting direction of the tool for automatic tool measurement with a rotating tool: ■ -: **M3** ■ +: **M4** **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1734 This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **-**, **+**

Parameter	Meaning
R-OFFS 	**Tool offset: radius?** Position of tool upon length measurement, offset between the center of the tool touch probe and the tool center for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1734 This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **-99999.9999...+99999.9999**
L-OFFS 	**Tool offset: length?** Position of tool upon radius measurement, distance between the top edge of the tool touch probe and the tool tip for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1734 Is added to the machine parameter **offsetToolAxis** (no. 122707) This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **-99999.9999...+99999.9999**
LBREAK 	**Breakage tolerance: length?** Permitted tool length deviation in breakage detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1734 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0...3.2767**
RBREAK 	**Breakage tolerance: radius?** Permitted tool radius deviation in breakage detection for automatic tool measurement. **Further information:** "Touch Probe Cycles: Automatic Tool Measurement", Page 1734 If the entered value is exceeded, the control locks the tool in column **TL**. This parameter applies to the following tools, regardless of technology: ■ Milling and drilling tools ■ Turning tools (option 50) Input: **0.0000...0.9999**
NMAX 	**Maximum speed [rpm]** Limitation of spindle speed for the programmed value including control by the potentiometer. Input: **0...999999**

Parameter	Meaning
LIFTOFF	**Lift-off allowed?** Automatic tool lift-off with active **M148** or allow **FUNCTION LIFTOFF**: ■ **Y**: Activate **LIFTOFF** ■ **N**: Deactivate **LIFTOFF** **Further information:** "Automatically lifting off upon an NC stop or a power failure with M148", Page 1257 **Further information:** "Automatic tool liftoff with FUNCTION LIFTOFF", Page 1110 Selection by means of a selection window Input: **Y**, **N**
TP_NO	**Number of the touch probe** Number of touch probe in the touch probe table **tchprobe.tp** **Further information:** "Touch probe table tchprobe.tp", Page 1839 Input: **0...99**
T-ANGLE 	**Point angle** Point angle of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of e.g. drills. **Further information:** "Cycles for milling", Page 452 Input: **-180...+180**
LAST_USE	**Date/time of last tool call** Time of last tool presence in the spindle This parameter applies to all tools, regardless of technology. Input: **00:00:00 01.01.1971**...**23:59:59 31.12.2030**
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table **Further information:** "Pocket table tool_p.tch", Page 1843 Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99**
AFC	**Feedback-control strategy** Control setting for adaptive feed control (AFC) (option 45) from the **AFC.tab** table. **Further information:** "Adaptive Feed Control (AFC, option 45)", Page 1114 Selection by means of a selection window Entry: **Text width 10**
ACC	**ACC active?** Activate or deactivate active chatter control (ACC) (option 145): ■ **Y**: Activate ■ **N**: Deactivate **Further information:** "Active Chatter Control (ACC, option 145)", Page 1121 Selection by means of a selection window Input: **Y**, **N**

Parameter	Meaning
PITCH 	**Tool thread pitch?** Thread pitch of the tool for automatic calculations within cycles. A positive sign means a right-hand thread. **Further information:** "Cycles for milling", Page 452 Input: **-9.9999...+9.9999**
AFC-LOAD	**Reference power for AFC [%]** Tool-dependent reference power for AFC (option 45). The input in percent refers to the rated spindle power. The control immediately uses the value given for feedback control, meaning a teach-in cut is dropped. Calculate the value beforehand with a teach-in step. **Further information:** "AFC teach-in cut", Page 1119 Input: **1.0...100.0**
AFC-OVLD1	**AFC overload warning level [%]** Cut-related tool wear monitoring for AFC (option 45). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. **Further information:** "Monitoring tool wear and tool load", Page 1120 Input: **0.0...100.0**
AFC-OVL2	**AFC ovrload switch-off level [%]** Cut-related tool load monitoring for AFC (option 45). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. **Further information:** "Monitoring tool wear and tool load", Page 1120 Input: **0.0...100.0**
KINEMATIC	**Tool-carrier kinematics** Assigning a tool carrier for exact definition of the tool for graphical simulation and collision monitoring. **Further information:** "Tool carrier management", Page 274 Selection by means of a selection window This parameter applies to all tools, regardless of technology. Entry: **Text width 20**
DR2TABLE	**Compensation val. table for DR2** Assigning a compensation value table ***.3dtc** for 3D tool radius compensation depending on the contact angle (option 92). This allows the control to compensate for, e.g., inaccuracies in the shape of a ball-nose cutter or the deflection behavior of a touch probe. **Further information:** "3D radius compensation depending on the tool contact angle (option 92)", Page 1062 Selection by means of a selection window Entry: **Text width 16**

Parameter	Meaning
OVRTIME	**Tool life expired** Time in minutes during which the tool may be used beyond the tool life defined in column **TIME1**. The machine manufacturer defines the function of this parameter. The machine manufacturer defines how the control uses the parameter when searching for tool names. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: **0...99**
RCUTS	**Width of the indexable insert** Front-face width of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring, e.g. for indexable inserts. Input: **0...99999.9999**

Notes

- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 1842
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
  - Tool that is in the spindle
  - Tool that is in the magazine

    > Refer to your machine manual.
    > If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.
  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with least remaining tool life.
- In the machine parameter **offsetToolAxis** (no. 122707), the machine manufacturer defines the distance between the upper edge of the tool touch probe and the tool tip.

  The parameter **L-OFFS** is added to this defined distance.
- In the machine parameter **zeroCutToolMeasure** (no. 122724) the machine manufacturer defines whether the control takes the parameter **R-OFFS** into account for automatic tool measurement.

33.5.3 Turning tool table toolturn.trn (option 50)

Application

The turning tool table **toolturn.trn** contains the data specific to turning tools.

Related topics

- Editing tool data in tool management
  Further information: "Tool management ", Page 270
- Tool data required for turning tools
  Further information: "Tool data for turning tools (option 50)", Page 261
- Milling-turning operations on the control
  Further information: "Turning (option 50)", Page 212
- General tool data, regardless of the technology
  Further information: "Tool table tool.t", Page 1814

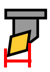
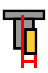
Requirements

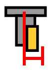
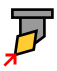
- Software option 50: Combined milling-turning
- Turning tool is defined in **TYPE** column of tool management
  Further information: "Tool types", Page 255

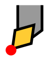
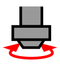
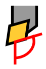
Description of function

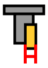
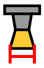
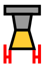
The file name of the turning tool table is **toolturn.trn** and this table must be stored in the folder **TNC:\table**.

The **toolturn.trn** turning tool table provides the following parameters:

Parameter	Meaning
T	Row number in the turning tool table The tool number allows identifying each tool unambiguously, e.g. for calling a tool. **Further information:** "Tool call by TOOL CALL", Page 277 You can define an index after the period. **Further information:** "Indexed tool", Page 250 The row number must match the tool number in the **tool.t** tool table. Input: **0.0...32767.9**
NAME	**Tool name?** The tool name allows identifying a tool, e.g. for calling a tool. **Further information:** "Tool call by TOOL CALL", Page 277 You can define an index after the period. **Further information:** "Indexed tool", Page 250 Input: **Text width 32**
ZL 	**Tool length 1?** Length of the tool in the Z direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 245 Input: **-99999.9999...+99999.9999**
XL 	**Tool length 2?** Length of the tool in the X direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 245 Input: **-99999.9999...+99999.9999**
YL 	**Tool length 3?** Length of the tool in the Y direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 245 Input: **-99999.9999...+99999.9999**

Parameter	Meaning
DZL 	**Oversize in tool length 1?** Delta value of tool length 1 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1449 Is added to the parameter **ZL** Input: **-99999.9999...+99999.9999**
DXL 	**Oversize in tool length 2?** Delta value of tool length 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1449 Is added to the parameter **XL** Input: **-99999.9999...+99999.9999**
DYL 	**Tool length oversize 3?** Delta value of tool length 3 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1449 Is added to the parameter **YL** Input: **-99999.9999...+99999.9999**
RS 	**Cutting edge radius?** The control takes into account the cutter radius for tool tip radius compensation. **Further information:** "Tooth radius compensation for turning tools (option 50)", Page 1041 In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** "Cycles for milling and turning", Page 677 For the cutter geometry, the control also considers the parameters **TO**, **T-ANGLE**, and **P-ANGLE**. Input: **0...99999.9999**
DRS 	**Cutter radius oversize?** Delta value of cutter radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1449 Is added to the parameter **RS** Input: **-999.9999...+999.9999**

Parameter	Meaning
TO 	**Tool orientation?** From the tool orientation, the control determines the position of the tool tip and, depending on the selected tool type, additional information such as the tool angle direction. This information is necessary, for example, for calculating the cutter radius compensation, milling cutter radius compensation, plunge angle, etc.. In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** "Cycles for milling and turning", Page 677 For the cutter geometry, the control also considers the parameters **RS**, **T-ANGLE**, and **P-ANGLE**. Input: **1...19**
SPB-INSERT 	**Angular offset?** Angular offset for recessing tools Input: **-90.0...+90.0**
ORI 	**Angle of spindle orientation?** Angle of tool spindle for aligning the turning tool Input: **-360.000...+360.000**
T-ANGLE 	**Tool angle** In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** "Cycles for milling and turning", Page 677 For the cutter geometry, the control also considers the parameters **RS**, **TO**, and **P-ANGLE**. Input: **0...179.999**
P-ANGLE 	**Point angle** In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. **Further information:** "Cycles for milling and turning", Page 677 For the cutter geometry, the control also considers the parameters **RS**, **TO**, and **T-ANGLE**. Input: **0...179.999**
CUTLENGTH  	**Cutting length of recessing tool** Length of the cutting edge of a turning or recessing tool The control monitors the length of the cutting edge in the turning cycles. If the cutting depth programmed in the turning cycle is greater than the length of the cutting edge defined in the tool table, then the control will display a warning and will automatically reduce the cutting depth. **Further information:** "Fundamentals of turning cycles", Page 693 Input: **0...99999.9999**

Parameter	Meaning
CUTWIDTH  	**Width of recessing tool** The control uses the width of a recessing tool for calculations within cycles. **Further information:** "Cycles for milling and turning", Page 677 Input: **0...99999.9999**
DCW 	**Oversize f. recessing tool width** Delta value of recessing tool width as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. **Further information:** "Programmable Touch Probe Cycles", Page 1449 Is added to parameter **CUTWIDTH** Input: **-99999.9999...+99999.9999**
TYPE 	**Type of turning tool** Depending on the selected turning tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Types within the turning tools", Page 256 **Further information:** "Tool management ", Page 270 Selection by means of a selection window Input: **ROUGH**, **FINISH**, **THREAD**, **RECESS**, **BUTTON**, and **RECTURN**
WPL-DX-DIAM	**Compensation value for the workpiece diameter** Compensation value for the workpiece diameter with respect to the working plane coordinate system (**WPL CS**). **Further information:** "Working plane coordinate system WPL-CS", Page 942 Input: **-99999.9999...+99999.9999**
WPL-DZL	**Compensation value for the workpiece length** Compensation value for the workpiece length with respect to the working plane coordinate system (**WPL CS**). **Further information:** "Working plane coordinate system WPL-CS", Page 942 Input: **-99999.9999...+99999.9999**

Notes

- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.
- Geometry values from the tool table **tool.t**, such as length **L** or radius **R**, are not effective with turning tools.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:

  - Tool that is in the spindle
  - Tool that is in the magazine

    > Refer to your machine manual.
    > If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with least remaining tool life.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 1842
- The columns **WPL-DX-DIAM** and **WPL-DZL** are deactivated in the default configuration.

  In the machine parameter **columnKeys** (no. 105501), the machine manufacturer activates the columns **WPL-DX-DIAM** and **WPL-DZL**. The names of the columns may be different, however.

33.5.4 Grinding tool table toolgrind.grd (option 156)

Application

The grinding tool table **toolgrind.grd** contains the data specific to grinding tools.

Related topics

- Editing tool data in tool management

  Further information: "Tool management ", Page 270
- Tool data required for grinding tools

  Further information: "Tool data for grinding tools (option 156)", Page 263
- Grinding operations on milling machines

  Further information: "Grinding operations (option 156)", Page 224
- Tool table for dressing tools

  Further information: "Dressing tool table tooldress.drs (option 156)", Page 1836
- General tool data, regardless of the technology

  Further information: "Tool table tool.t", Page 1814

Requirements

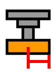
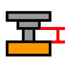
- Software option 156: Jig grinding
- Grinding tool is defined in **TYPE** column of tool management

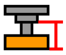
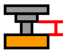
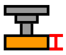
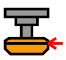
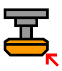
  Further information: "Tool types", Page 255

Description of function

The file name of the grinding tool table is **toolgrind.grd** and this table must be stored in the folder **TNC:\table**.

The **toolgrind.grd** grinding tool table provides the following parameters:

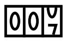
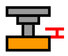
Parameter	Meaning
T	**Tool number** Row number in the grinding tool table The tool number allows identifying each tool unambiguously, e.g. for calling a tool. **Further information:** "Tool call by TOOL CALL", Page 277 You can define an index after the period. **Further information:** "Indexed tool", Page 250 The row number must match the tool number in the **tool.t** tool table Input: **0...32767**
NAME	**Name of grinding wheel** The tool name allows identifying a tool, e.g. for calling a tool. **Further information:** "Tool call by TOOL CALL", Page 277 You can define an index after the period. **Further information:** "Indexed tool", Page 250 Input: **Text width 32**
TYPE 	**Type of grinding wheel** Depending on the selected grinding tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Types within the grinding tools", Page 257 **Further information:** "Tool management ", Page 270 Selection by means of a selection window Input: **GRIND_M**, **GRIND_MS**, **GRIND_MT**, **GRIND_S**, **GRIND_A**, and **GRIND_P**
R-OVR 	**Radius of grinding wheel** Outermost radius of grinding tool After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.000000...999.999999**
L-OVR 	**Overhang of grinding wheel** Length up to the outermost radius of the grinding tool, with respect to the tool carrier preset After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.000000...999.999999**

Parameter	Meaning
LO 	**Overall length** Absolute length of the grinding tool, with respect to the tool carrier preset After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.000000...999.999999**
LI 	**Length to the inner edge** Length up to the inner edge, with respect to the tool carrier preset After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.000000...999.999999**
B 	**Width** Width of the grinding tool After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.000000...999.999999**
G 	**Depth** Depth of grinding wheel After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.000000...999.999999**
ALPHA	**Angle for the slant** After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.00000...90.00000**
GAMMA	**Angle for the corner** After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **45.00000...180.00000**
RV 	**Radius at the edge for L-OVR** After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.00000...999.99999**
RV1 	**Radius at the edge for LO** After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.00000...999.99999**

Parameter	Meaning
RV2	**Radius at the edge for LI** After initial dressing, you can no longer edit this parameter. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0.00000...999.99999**
dR-OVR	**Compensation of the radius** Delta value of the radius for tool compensation The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter **R-OVR** Input: **-999.999999...+999.999999**
dL-OVR	**Compensation of the overhang** Delta value of the overhang for tool compensation The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter **L-OVR** Input: **-999.999999...+999.999999**
dLO	**Compensation of the total length** Delta value of the total length for tool compensation The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter **LO** Input: **-999.999999...+999.999999**
dLI	**Compensation of the length to the inner edge** Delta value of the length up to the inner edge for tool compensation The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter **LI** Input: **-999.999999...+999.999999**
R_SHAFT	**Radius of the tool shank** Input: **0.00000...999.99999**
R_MIN	**Min. permissible radius** If, after dressing, the actual radius is below the minimum permissible radius defined here, the control will display an error message. Input: **0.00000...999.99999**
B_MIN	**Min. permissible width** If, after dressing, the actual width is below the minimum permissible width defined here, the control will display an error message. Input: **0.00000...999.99999**

Parameter	Meaning
V_MAX 	**Maximum permissible cutting speed** Cutting speed limit This value cannot be exceeded by programming a higher value or by using the potentiometer. Input: **0.000...999.999**
V	**Current cutting speed** Currently no function Input: **0.000...999.999**
W	**Tilt angle** Currently no function Input: **-90.00000...90.0000**
W_TYPE	**Tilted toward inner or outer edge** Currently no function Input: **-1**, **0**, **+1**
KIND	**Type of machining (internal/external grinding)** Currently no function Input: **0**, **1**
HW	**Wheel has a relief cut** Grinding wheel with or without relief cut: ▪ **0**: **No relief cut** ▪ **1**: **With relief cut** Selection by means of a selection window Input: **0**, **1**
HWA 	**Angle for relief cut on the outer edge** Input: **0.00000...45.00000**
HWI 	**Angle for relief cut on the inner edge** Input: **0.00000...45.00000**
INIT_D_OK	**Initial dressing performed** Initial dressing is the first dressing operation performed on the grinding wheel. If the following requirements are fulfilled, the control will set the parameter **INIT_D_OK** to **1**: ▪ Grinding tool is defined ▪ Initial dressing performed If the parameter **INIT_D_OK** is set to **1**, the control will disable the parameters for defining the grinding tool. If you set the parameter **INIT_D_OK** to **0**, the control will re-enable the editing of the parameters. In this case, the control will have to perform initial dressing of the tool again. **Further information:** "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889 Input: **0**, **1**

Parameter	Meaning
INIT_D_PNR	**Dresser location for initial dressing** Dressing location used for initial dressing Input: **0...9999**
INIT_D_DNR	**Dresser number for initial dressing** Number of the dresser used for initial dressing Input: **0...32767**
MESS_OK	**Measure the grinding wheel** Currently no function Input: **0**, **1**
STATE	**Setup status** Currently no function Input: **%0000000000000000...%1111111111111111**
A_NR_D	**Dresser number (diameter dressing)** Number of dresser used for dressing the diameter Input: **0...32767**
A_NR_A	**Dresser number (outer edge dressing)** Number of dresser used for dressing the outer edge Input: **0...32767**
A_NR_I	**Dresser number (inner edge dressing)** Number of dresser used for dressing the inner edge Input: **0...32767**
DRESS_N_D	**Dressing counter for diameter (specification)** Specified number of dressing cycle calls that will be skipped until the next dressing of the diameter. Input: **0...999**
DRESS_N_A	**Dressing counter for outer edge (specification)** Specified number of dressing cycle calls that will be skipped until the next dressing of the outer edge. Input: **0...999**
DRESS_N_I	**Dressing counter for inner edge (specification)** Specified number of dressing cycle calls that will be skipped until the next dressing of the inner edge. Input: **0...999**
DRESS_N_D_ACT	**Current dressing counter of the diameter** Current number of dressing cycles that have been skipped since the last dressing of the diameter. Input: **0...999**
DRESS_N_A_ACT	**Current dressing counter of the outer edge** Current number of dressing cycles that have been skipped since the last dressing of the outer edge. Input: **0...999**

Parameter	Meaning
DRESS_N_I_ACT 	**Current dressing counter of the inner edge** Current number of dressing cycles that have been skipped since the last dressing of the inner edge. Input: **0...999**
AD 	**Retraction amount at the diameter** The control uses this parameter when using a cycle for dressing. **Further information:** "General information on the dressing cycles", Page 843 Input: **0.00000...999.99999**
AA 	**Retraction amount at the outer edge** The control uses this parameter when using a cycle for dressing. **Further information:** "General information on the dressing cycles", Page 843 Input: **0.00000...999.99999**
AI 	**Retraction amount at the inner edge** The control uses this parameter when using a cycle for dressing. **Further information:** "General information on the dressing cycles", Page 843 Input: **0.00000...999.99999**
FORM	**Wheel shape** Currently no function Input: **0.00...99.99**
A_PL	**Chamfer length at outside** Currently no function Input: **0.00000...999.99999**
A_PW	**Chamfer angle at outside** Currently no function Input: **0.00000...89.99999**
A_R1	**Corner radius at outside** Currently no function Input: **0.00000...999.99999**
A_L	**Length of outside** Currently no function Input: **0.00000...999.99999**
A_HL	**Length of relief cut, wheel depth at outside** Currently no function Input: **0.00000...999.99999**
A_HW	**Angle of relief cut at outside** Currently no function Input: **0.00000...45.00000**
A_S	**Side depth at outside** Currently no function Input: **0.00000...999.99999**
A_R2	**Angle of departure at outside** Currently no function Input: **0.00000...999.99999**

Parameter	Meaning
A_G	**Reserve at outside** Currently no function Input: **0.00000...999.99999**
I_PL	**Chamfer length at inside** Currently no function Input: **0.00000...999.99999**
I_PW	**Chamfer angle at inside** Currently no function Input: **0.00000...89.99999**
I_R1	**Corner radius at inside** Currently no function Input: **0.00000...999.99999**
I_L	**Length of inside** Currently no function Input: **0.00000...999.99999**
I_HL	**Length of relief cut, wheel depth at inside** Currently no function Input: **0.00000...999.99999**
I_HW	**Angle of relief cut at inside** Currently no function Input: **0.00000...45.00000**
I_S	**Side depth at inside** Currently no function Input: **0.00000...999.99999**
I_R2	**Angle of departure at inside** Currently no function Input: **0.00000...999.99999**
I_G	**Reserve at inside** Currently no function Input: **0.00000...999.99999**

Notes

- Geometry values from the tool table **tool.t**, such as length or radius, are not effective with grinding tools.
- When dressing a grinding tool, the tool must not be assigned a tool carrier kinematic model.
- Measure the grinding tool after dressing so that the control enters the correct delta values.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
  - Tool that is in the spindle
  - Tool that is in the magazine

    > Refer to your machine manual.
    > If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with least remaining tool life.
- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 1842

33.5.5 Dressing tool table tooldress.drs (option 156)

Application

The dressing tool table **tooldress.drs** contains the data specific to dressing tools.

Related topics

- Editing tool data in tool management

  Further information: "Tool management ", Page 270
- Tool data required for dressing tools

  Further information: "Tool data for dressing tools (option 156)", Page 267
- Initial dressing

  Further information: "Cycle 1032 GRINDING WHL LENGTH COMPENSATION (option 156)", Page 889
- Grinding operations on milling machines

  "Grinding operations (option 156)"
- Tool table for grinding tools

  Further information: "Grinding tool table toolgrind.grd (option 156)", Page 1828
- General tool data, regardless of the technology

  Further information: "Tool table tool.t", Page 1814

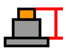
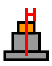
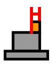
Requirements

- Software option 156: Jig grinding
- Dressing tool is defined in **TYPE** column of tool management
  Further information: "Tool types", Page 255

Description of function

The file name of the dressing tool table is **tooldress.drs** and this table must be stored in the folder **TNC:\table**.

The **tooldress.drs** dressing tool table provides the following parameters:

Parameter	Meaning
T	Row number in the dressing tool table The tool number allows identifying each tool unambiguously, e.g. for calling a tool. **Further information:** "Tool call by TOOL CALL", Page 277 You can define an index after the period. **Further information:** "Indexed tool", Page 250 The row number must match the tool number in the **tool.t** tool table. Input: **0.0...32767.9**
NAME	**Name of dressing tool** The tool name allows identifying a tool, e.g. for calling a tool. **Further information:** "Tool call by TOOL CALL", Page 277 You can define an index after the period. **Further information:** "Indexed tool", Page 250 Input: **Text width 32**
ZL 	**Tool length 1** Length of the tool in the Z direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 245 Input: **-99999.9999...+99999.9999**
XL 	**Tool length 2** Length of the tool in the X direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 245 Input: **-99999.9999...+99999.9999**
YL 	**Tool length 3** Length of the tool in the Y direction, with respect to the tool carrier preset **Further information:** "Tool carrier reference point", Page 245 Input: **-99999.9999...+99999.9999**
DZL 	**Tool length oversize 1** Delta value of tool length 1 for tool compensation Is added to the parameter **ZL** Input: **-99999.9999...+99999.9999**
DXL 	**Tool length oversize 2** Delta value of tool length 2 for tool compensation Is added to the parameter **XL** Input: **-99999.9999...+99999.9999**

Parameter	Meaning
DYL 	**Tool length oversize 3** Delta value of tool length 3 for tool compensation Is added to the parameter **YL** Input: **-99999.9999...+99999.9999**
RS 	**Tool tip radius** Input: **0.0000...99999.9999**
DRS 	**Cutter radius oversize** Delta value of the cutter radius for tool compensation Is added to the parameter **RS** Input: **-999.9999...+999.9999**
TO 	**Tool orientation** The control uses the tool orientation to determine the position of the tool's cutting edge. Input: **1...9**
CUTWIDTH	**Width of tool (plate, roll)** Tool width of the tool types **dressing plate** and **dressing roll** Input: **0.0000...99999.9999**
TYPE 	**Type of dressing tool** Depending on the selected dressing tool type, the control displays the suitable tool parameters in the **Form** workspace of tool management. **Further information:** "Types within the dressing tools", Page 257 **Further information:** "Tool management ", Page 270 Selection by means of a selection window Input: **DIAMOND**, **SPINDLE**, **PLATE**, and **ROLL**
N-DRESS	**Speed of the tool (dressing spindle)** Shaft speed of a dressing spindle or dressing roll Input: **0.0000...99999.9999**

Notes

- When dressing a grinding tool, the tool must not be assigned a tool carrier kinematic model.
- Geometry values from the tool table **tool.t**, such as length or radius, are not effective with dressing tools.
- Assign unique tool names!

  If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
  - Tool that is in the spindle
  - Tool that is in the magazine

    > Refer to your machine manual.
    > If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

  - Tool that is defined in the tool table but is currently not in the magazine

  If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with least remaining tool life.
- If you want to archive tool tables, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

  Further information: "Creating a tool table in inches", Page 1842

33.5.6 Touch probe table tchprobe.tp

Application

The touch probe table **tchprobe.tp** defines the touch probe and data for the probing process, e.g. the probing feed rate. If you use several touch probes, you can save separate data for each touch probe.

Related topics

- Editing tool data in tool management

  Further information: "Tool management ", Page 270
- Touch probe functions

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1425
- Programmable touch probe cycles

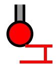
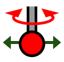
  Further information: "Programmable Touch Probe Cycles", Page 1449

Description of function

The file name of the touch probe table is **tchprobe.tp** and this table must be stored in the folder **TNC:\table**.

The touch probe table **tchprobe.tp** provides the following parameters:

Parameter	Meaning
NO	**Sequential number of touch probe** You use this number to assign the touch probe to the data in the tool management column **TP_NO**. Input: **1...99**
TYPE	**Selection of the touch probe?** The TS 642 touch probe makes the following values available: ■ **TS642-3**: The touch probe is activated by a conical switch. This mode is not supported. ■ **TS642-6**: The touch probe is activated by an infrared signal. Select this mode. Input: **TS120**, **TS220**, **TS249**, **TS260**, **TS440**, **TS444**, **TS460**, **TS630**, **TS632**, **TS640**, **TS642-3**, **TS642-6**, **TS649**, **TS740**, **TS 760**, **KT130**, **OEM**
CAL_OF1	**TS center misalignmt. ref. axis? [mm]** Offset of the touch probe axis to the spindle axis in the main axis Input: **-99999.9999...+99999.9999**
CAL_OF2	**TS center misalignmt. aux. axis? [mm]** Offset of the touch probe axis to the spindle axis in the secondary axis Input: **-99999.9999...+99999.9999**
CAL_ANG	**Spindle angle for calibration?** Prior to calibrating or probing, the control aligns the touch probe with the spindle orientation angle (if possible). Input: **0.0000...359.9999**
F	**Probing feed rate? [mm/min]** In the machine parameter **maxTouchFeed** (no. 122602), the machine manufacturer defines the maximum probing feed rate. If **F** is greater than the maximum probing feed rate, then the maximum probing feed rate will be used. Input: **0...9999**
FMAX	**Rapid traverse in probing cycle? [mm/min]** Feed rate at which the control pre-positions the touch probe and positions it between the measuring points Input: **-99999...+99999**
DIST	**Maximum measuring range? [mm]** If the stylus is not deflected in a probing process within the defined value, the control will display an error message. Input: **0.00100...99999.99999**

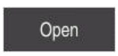
Parameter	Meaning
SET_UP 	**Set-up clearance? [mm]** Distance of touch probe from the defined touch point when pre-positioning The smaller this value is, the more exactly you must define the touch point position. Safety clearances defined in the touch probe cycle are added to this value. Input: **0.00100...99999.99999**
F_PREPOS 	**Pre-position at rapid? ENT/NOENT** Speed for pre-positioning: ■ **FMAX_PROBE**: Pre-position at the speed from **FMAX** ■ **FMAX_MACHINE**: Pre-position at machine rapid traverse Input: **FMAX_PROBE**, **FMAX_MACHINE**
TRACK 	**Probe oriented? Yes=ENT/No=NOENT** Orienting the infrared touch probe in each probing process: ■ **ON**: The control orients the touch probe in the defined probing direction. In this way, the stylus is always deflected in the same direction, improving measuring accuracy. ■ **OFF**: The control will not orient the touch probe. If you change the **TRACK** parameter, you must recalibrate the touch probe. Input: **ON**, **OFF**
SERIAL 	**Serial number?** The control automatically edits this parameter of touch probes with an EnDat interface. Input: **Text width 15**
REACTION	**Reaction? EMERGSTOP=ENT/NCSTOP=NOENT** As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. Reaction to resetting the ready signal: ■ **NCSTOP**: Interrupt NC program ■ **EMERGSTOP**: Emergency stop, quick braking of the axes Input: **NCSTOP**, **EMERGSTOP**

Editing the touch probe table

To edit the touch probe table:

- ▶ Select the **Tables** operating mode

- ▶ Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- ▶ Select the **tchprobe.tp** file in the **Open File** workspace



- ▶ Select **Open**
- > The control opens the **Touch probes** application.

- ▶ Activate **Edit**
- ▶ Select the desired value
- ▶ Edit the value

Notes

- You can also edit the touch probe table values in the tool management.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- In the machine parameter **overrideForMeasure** (no. 122604), the machine manufacturer defines whether you will be allowed to change the feed rate with the feed-rate potentiometer during probing.

33.5.7 Creating a tool table in inches

To create a tool table in inches:

- ▶ Select the **Manual** operating mode

T

- ▶ Select **T**
- ▶ Select the tool **T0**
- ▶ Press the **NC Start** key
- > The control removes the current tool and does not insert a new tool.
- ▶ Restart the control
- ▶ Do not acknowledge **Power interrupted**
- ▶ Select the **Files** operating mode
- ▶ Open the **TNC:\table** folder
- ▶ Rename the original file (e.g., **tool.t** as **tool_mm.t**)
- ▶ Select the **Tables** operating mode
- ▶ Select **Add**
- ▶ Select **Create new table**
- > The control opens the **Create new table** window.
- ▶ Select a folder with a corresponding file extension (e.g., **t**)

Select a path

- ▶ Select **Select a path**
- > The control opens the **Save as** window.
- ▶ Select the **table** folder
- ▶ Enter a name (e.g., **tool**)

Create

- ▶ Select **Create**

OK

- ▶ Select **OK**
- > The control opens the **Tool table** tab in **Tables** operating mode.
- ▶ Restart the control

CE

- ▶ Acknowledge **Power interrupted** with the **CE** key
- ▶ Select the **Tool table** tab in **Tables** operating mode
- > The control uses the newly created table as a tool table.

33.6 Pocket table tool_p.tch

Application

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine. The control needs the pocket table in order to change the tool.

Related topics

- Tool call

  Further information: "Tool call", Page 277
- Tool table

  Further information: "Tool table tool.t", Page 1814

Requirement

- The tool is defined in the tool management.

  Further information: "Tool management ", Page 270

Description of function

The file name of the pocket table is **tool_p.tch** and this table must be stored in the folder **TNC:\table**.

The **tool_p.tch** pocket table provides the following parameters:

Parameter	Meaning
P	**Pocket number?** Pocket number of the tool in the tool magazine Input: **0.0...99.9999**
T	**Tool number?** Row number of the tool from the tool table **Further information:** "Tool table tool.t", Page 1814 Input: **1...99999**
TNAME	**Tool name?** Name of the tool from the tool table When you define the tool number, the control will automatically load the tool name. **Further information:** "Tool table tool.t", Page 1814 Input: **Text width 32**
RSV	**Reserve pocket?** When a tool is in the spindle, the control reserves the pocket of this tool in the box magazine. To reserve the pocket for the tool: ■ No value entered: Pocket is not reserved ■ **R**: Pocket is reserved Input: No value, **R**
ST	**Special tool?** Define the tool as a special tool (e.g., with oversize tools): ■ No value entered: No special tool ■ **S**: Special tool Input: No value, **S**

Parameter	Meaning
F	**Fixed pocket?** Always return the tool to the same pocket in the tool magazine (e.g., with special tools) To define a fixed pocket for the tool: ■ No value entered: No fixed pocket ■ **F**: Fixed pocket Input: No value, **F**
L	**Locked pocket?** To lock a pocket for tools (e.g., the pockets next to special tools): ■ No value entered: Do not lock ■ **L**: Lock Input: No value, **L**
DOC	**Pocket comment?** The control automatically loads the tool comment from the tool table. **Further information:** "Tool table tool.t", Page 1814 Input: **Text width 32**
PLC	**PLC status?** Information about this tool pocket, which is transferred to the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **%00000000...%11111111**
P1 ... P5	**Value?** The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: **-99999.9999...+99999.9999**
PTYP	**Tool type for pocket table?** Tool type for evaluation in the pocket table The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: **0...99**
LOCKED_ABOVE	**Lock pocket above?** Box magazine: Lock the pocket above This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_BELOW	**Lock pocket below?** Box magazine: Lock the pocket below This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LOCKED_LEFT	**Lock pocket at left?** Box magazine: Lock the pocket at left This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**

Parameter	Meaning
LOCKED_RIGHT	**Lock pocket at right?** Box magazine: Lock the pocket at right This parameter depends on the machine. Refer to your machine manual. Input: **0...99999**
LAST_USE	**LAST_USE** The control automatically loads the date and time of the last tool call from the tool table. **Further information:** "Tool table tool.t", Page 1814 Refer to your machine manual. Entry: **Text width 20**
S1	**S1** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **Text width 16**
S2	**S2** Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: **Text width 16**

33.7 Tool usage file

Application

The control saves information about the tools of an NC program in a tool usage file (e.g., all the required tools and the tool usage times). The control needs this file for the tool usage test.

Related topics

- Using the tool usage test
  Further information: "Tool usage test", Page 284
- Working with pallet tables
  Further information: "Pallet Machining and Job Lists", Page 1763
- Tool data from the tool table
  Further information: "Tool table tool.t", Page 1814

Requirements

- **Generate tool-usage file** is enabled by your machine manufacturer
  The machine manufacturer uses the machine parameter **createUsageFile** (no. 118701) to define whether the **Generate tool-usage file** function is enabled.
  Further information: "Creating the tool usage file", Page 285
- The **Generate tool-usage file** function setting is set to either **once** or **always**
  Further information: "Channel settings", Page 1915

Description of function

The tool usage file provides the following parameters:

Parameter	Meaning
NR	Row number in the tool usage file Input: **0...99999**
TOKEN	In the **TOKEN** column, the control uses one word to show which information is contained in the respective row: ■ **TOOL**: Data per tool call; listed in chronological order ■ **TTOTAL**: All data of a tool; listed in alphabetical order ■ **STOTAL**: Called NC programs; listed in chronological order ■ **TIMETOTAL**: Total tool usage time of an NC program ■ **TOOLFILE**: Path of the tool table This enables the control during the tool usage test to detect whether you have performed the simulation with the tool table **tool.t** Input: **Text width 17**
TNR	Tool number If the control has not yet inserted a tool, the column contains the value **-1**. Input: **-1...32767**
IDX	Tool index Input: **0...9**
NAME	Tool name Input: **Text width 32**
TIME	Tool usage time in seconds Time during which the tool is cutting a workpiece (excluding rapid traverse movements) Input: **0...9999999**
WTIME	Total tool usage time in seconds Total time between the tool changes, during which the tool is cutting a workpiece Input: **0...9999999**
RAD	Sum of the tool radius **R** and the delta radius **DR** from the tool table Input: **-999999.9999...999999.9999**
BLOCK	NC block number of the tool call Input: **0...999999999**
PATH	Path of the NC program, the pallet table, or the tool table Input: **Text width 300**
T	Tool number, including the tool index If the control has not yet inserted a tool, the column contains the value **-1**. Input: **-1...32767.9**

Parameter	Meaning
OVRMAX	Maximum feed-rate override If you only simulate the machining operation, then the control will enter the value **100**. Input: **0...32767**
OVRMIN	Minimum feed rate override If you only simulate the machining operation, then the control will enter the value **-1**. Input: **-1...32767**
NAMEPRG	Type of tool definition during a tool call: ■ **0**: The tool number is programmed ■ **1**: The tool name is programmed Input: **0**, **1**
LINENR	Row number of the pallet table in which the NC program is defined Input: **-1...99999**

Note

The control creates dependency files (***.dep**); for example, the tool-usage file in order to perform a tool usage test.

In the machine parameter **dependentFiles** (no. 122101) the machine manufacturer defines whether the control displays dependency files.

33.8 T usage order (option 93)

Application

In the **T usage order** table, the control displays the tool call sequence in an NC program. Before starting the program, you can see, for example, when a manual tool change will take place.

Requirements

- Software option 93: Extended Tool Management
- Tool-usage file has been created

  Further information: "Creating the tool usage file", Page 285

  Further information: "Tool usage file", Page 1845

Description of function

When you select an NC program in **Program Run** operating mode, the control will automatically create the **T usage order** table. The control displays the table in the **T usage order** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in chronological order. You cannot edit the table.

The **T usage order** table provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows
T	Number of the tool used, including an index as needed **Further information:** "Indexed tool", Page 250 May differ from the programmed tool (e.g., when a replacement tool is used)
NAME	Name of the tool used, including an index as needed **Further information:** "Indexed tool", Page 250 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: ■ **OK**: Tool is in order ■ **Locked**: Tool is locked ■ **Not found**: Tool is not defined in the pocket table **Further information:** "Pocket table tool_p.tch", Page 1843 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool management ", Page 270
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 250
USAGE	Total tool usage time from the **WTIME** column of the **tool usage file** (in seconds) Total time between the tool changes, during which the tool is cutting a workpiece **Further information:** "Tool usage file", Page 1845
TOOL TIME	Estimated time of tool change
M3/M4 TIME	Tool usage time from the **TIME** column of the **tool usage file** (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) **Further information:** "Tool usage file", Page 1845
MIN OVRD	Minimum value of the feed-rate potentiometer during program run (in percent)
MAX OVRD	Maximum value of the feed-rate potentiometer during program run (in percent)
NC PGM	Path of the NC program in which the tool is programmed
MAGAZINE	In this column, the control writes whether the tool is currently in the magazine or in the spindle. This column remains empty if the tool is a zero tool or not defined in the pocket table. **Further information:** "Pocket table tool_p.tch", Page 1843

33.9 Tooling list (option 93)

Application

In the **Tooling list** table, the control displays information about all the tools called within an NC program. Before starting the program, you can check, for example, whether all tools are contained in the magazine.

Requirements

- Software option 93: Extended Tool Management
- Tool-usage file has been created

  Further information: "Creating the tool usage file", Page 285

  Further information: "Tool usage file", Page 1845

Description of function

When you select an NC program in **Program Run** operating mode, the control will automatically create the **Tooling list** table. The control displays the table in the **Tooling list** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in chronological order. You cannot edit the table.

The **Tooling list** table provides the following parameters:

Parameter	Meaning
T	Number of the tool used, including an index as needed **Further information:** "Indexed tool", Page 250 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: ■ **OK**: Tool is in order ■ **Locked**: Tool is locked ■ **Not found**: Tool is not defined in the pocket table **Further information:** "Pocket table tool_p.tch", Page 1843 ■ **T no. missing**: Tool is not defined in the tool management **Further information:** "Tool carrier management", Page 274
T PROG	Number or name of the programmed tool, including an index as needed **Further information:** "Indexed tool", Page 250
M3/M4 TIME	Tool usage time from the **TIME** column of the **tool usage file** (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) **Further information:** "Tool usage file", Page 1845
MAGAZINE	In this column, the control writes whether the tool is currently in the magazine or in the spindle. This column remains empty if the tool is a zero tool or not defined in the pocket table. **Further information:** "Pocket table tool_p.tch", Page 1843

33.10 Freely definable tables

Application

In freely definable tables you can save and read any information from the NC program. The Q parameter functions **FN 26** to **FN 28** are provided for this purpose.

Related topics

- Variable functions **FN 26** to **FN 28**

  Further information: "Functions for freely definable tables", Page 1296

Description of function

When you create a freely definable table, the control will provide various table templates for selection.

The machine manufacturers can create their own table templates and store them in the control.

33.10.1 Creating freely definable tables

To create a freely definable table:

- Select the **Tables** operating mode
- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the **tab** folder
- Select a table template

Select a path

- Select **Select a path**
- > The control opens the **Save as** window.
- Select a folder
- Enter a name

Create

- Select **Create**
- > The control opens the table.
- Modify the table as needed

  Further information: "Table workspace", Page 1803

Note

The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

Further information: "Table access with SQL statements", Page 1316

33.11 Preset table

Application

The **preset.pr** preset table allows you to manage presets, such as the position and misalignment of a workpiece in the machine. The active row in the preset table is used as a workpiece preset in the NC program and as the coordinate origin of the workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 187

Related topics

- Setting and activating presets

  Further information: "Preset management", Page 949

Description of function

The preset table is stored in the **TNC:\table** directory by default and is named **preset.pr**. In the **Tables** operating mode, the preset table is open by default.

> Refer to your machine manual.
> The machine tool builder can define a different path for the preset table.
> In the optional machine parameter **basisTrans** (no. 123903), the machine manufacturer defines a specific preset table for each range of traverse.

Preset table icons

The preset table contains the following icons:

Icon	Function
⌖	Active row
🔒	Write-protected row

Preset table parameters

The preset table contains the following parameters:

Parameter	Meaning
NO	Number of preset table row Input: **0...99999999**
DOC	Comment Entry: **Text width 16**
X	X coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 938 Input: **-99999.99999...+99999.99999**
Y	Y coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 938 Input: **-99999.99999...+99999.99999**

Parameter	Meaning
Z	Z coordinate of preset Basic transformation relating to the basic coordinate system **B-CS** **Further information:** "Basic coordinate system B-CS", Page 938 Input: **-99999.99999...+99999.99999**
SPA	Spatial angle of preset in the A axis Basic transformation relating to the basic coordinate system **B-CS**, the preset contains a 3D basic rotation in tool axis **Z**. **Further information:** "Basic coordinate system B-CS", Page 938 Input: **-99999.9999999...+99999.9999999**
SPB	Spatial angle of preset in the B axis Basic transformation relating to the basic coordinate system **B-CS**, the preset contains a 3D basic rotation in tool axis **Z**. **Further information:** "Basic coordinate system B-CS", Page 938 Input: **-99999.9999999...+99999.9999999**
SPC	Spatial angle of preset in the C axis Basic transformation relating to the basic coordinate system **B-CS**, the preset contains a basic rotation in tool axis **Z**. **Further information:** "Basic coordinate system B-CS", Page 938 Input: **-99999.9999999...+99999.9999999**
X_OFFS	Position of the X axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935 Input: **-99999.99999...+99999.99999**
Y_OFFS	Position of the Y axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935 Input: **-99999.99999...+99999.99999**
Z_OFFS	Position of the Z axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935 Input: **-99999.99999...+99999.99999**
A_OFFS	Axis angle of the A axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935 Input: **-99999.9999999...+99999.9999999**
B_OFFS	Axis angle of the B axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935 Input: **-99999.9999999...+99999.9999999**
C_OFFS	Axis angle of the C axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935 Input: **-99999.9999999...+99999.9999999**

Parameter	Meaning
U_OFFS	Position of the U axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935 Input: **-99999.99999...+99999.99999**
V_OFFS	Position of the V axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935 Input: **-99999.99999...+99999.99999**
W_OFFS	Position of the W axis for the preset Offset relating to the machine coordinate system **M-CS** **Further information:** "Machine coordinate system M-CS", Page 935 Input: **-99999.99999...+99999.99999**
ACTNO	Active workpiece preset The control automatically enters **1** in the active row. Input: **0**, **1**
LOCKED	Write-protection of the table row Entry: **Text width 16**

Refer to your machine manual.
In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.

Write-protection for table rows

The **LOCKED** column allows protecting any rows of the preset table against overwriting. You can protect the row with or without a password.

The control displays an icon at the beginning of write-protected rows.

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This locks the protected rows permanently.

- Protecting table rows without a password is recommended
- Note down your passwords

33.11.1 Activating write protection

Protecting table rows without a password

To protect a table row without a password:

- Activate the **Edit** switch
- Select the **LOCKED** column in the desired row
- Enter **L**
- Confirm your input
- > The control protects the row from being edited and displays an icon at the beginning of the row.

Protecting table rows with a password

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This locks the protected rows permanently.

- Protecting table rows without a password is recommended
- Note down your passwords

To protect a table row with a password:

- Activate the **Edit** switch
- Select the **LOCKED** column in the desired row
- Enter the password
- Confirm your input
- > The control displays the characters **###** in the **LOCKED** column.
- > The control protects the row from being edited and displays an icon at the beginning of the row.

33.11.2 Removing write protection

Unlocking table rows that are protected without a password

To unlock a table row that is protected without a password:

- Activate the **Edit** switch
- Select the **LOCKED** column in the desired row
- Enter **L** again
- Confirm your input
- > The control enables the row for editing and removes the icon at the beginning of the row.

Unlocking table rows that are protected with a password

> **NOTICE**
>
> **Caution: Data may be lost!**
>
> Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This locks the protected rows permanently.
>
> - Protecting table rows without a password is recommended
> - Note down your passwords

To unlock a table row that is protected with a password:

- Activate the **Edit** switch
- Select the **LOCKED** column in the desired row
- Delete **###**
- Enter the password
- Confirm your input
- > The control enables the row for editing and removes the icon at the beginning of the row.

33.11.3 Creating a preset table in inches

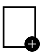
If you define inches as the unit of measure in the machine parameter **unitOfMeasure** (no. 101101), the unit of measure of the preset table will not be adjusted automatically.

To create a preset table in inches:

- Select the **Files** operating mode
- Open the **TNC:\table** folder
- Rename the **preset.pr** file (e.g., as **preset_inch.pr**)

- Select the **Tables** operating mode

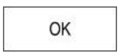
- Select **Add**



- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the **pr** folder

- Select the desired **prototype**
- > The control opens the **Save as** window.
- Select the **table** folder
- Enter the name **preset.pr**

- Select **Create**



- Select **OK**
- > The control opens the **Presets** tab in **Tables** operating mode.
- Restart the control

- Select the **Presets** tab in **Tables** operating mode
- > The control uses the newly created table as a preset table.

Notes

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

- Before activating a preset, check whether all columns contain values.

- To optimize the file size and the processing speed, keep the preset table as short as possible.
- New rows can be inserted only at the end of the preset table.
- The machine manufacturer uses the optional **initial** machine parameter (no. 105603) to define a default value for each column of a new row.
- If the unit of measure of the preset table is not identical with the unit of measure defined in the machine parameter **unitOfMeasure** (no. 101101), the control displays a message in the diagnostics bar when in the **Tables** operating mode.
- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.

  Further information: "Pallet preset table", Page 1775

33.12 Point table

Application

In a point table, you save randomly distributed points on a workpiece. The control calls a cycle at each point. You can hide individual points and define a clearance height.

Related topics

- Calling point tables, effect with different cycles

  Further information: "Point tables", Page 346

Description of function

Parameters in point tables

The point table provides the following parameters:

Parameter	Meaning
NR	Row number in the point table Input: **0...99999**
X	X coordinate of a point Input: **-99999.9999...+99999.9999**
Y	Y coordinate of a point Input: **-99999.9999...+99999.9999**
Z	Z coordinate of a point Input: **-99999.9999...+99999.9999**
FADE	**Hide? (yes=ENT/no=NO ENT)** **Y**=**Yes:** The point is hidden during machining. Points that have been hidden will remain hidden until they are manually shown again. **N**=**No:** The point is shown for machining. All points of a point table are shown for machining by default. Input: **Y**, **N**
CLEARANCE	**Clearance height?** Safe position in the tool axis to which the control retracts the tool after machining a point. If you do not define a value in the **CLEARANCE** column, the control will use the value of the cycle parameter **Q204 2ND SET-UP CLEARANCE**. If you have defined values in both the **CLEARANCE** column and the **Q204** parameter, the control will use the higher of the two values. Input: **-99999.9999...+99999.9999**

33.12.1 Creating a point table

To create a point table:

- Select the **Tables** operating mode
- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the **pnt** folder
- Select the desired **prototype**
- > The control opens the **Save as** window.
- Select a folder
- Enter name
- Select **Create**
- Select **OK**
- > The control opens the point table.

> The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.
> **Further information:** "Table access with SQL statements", Page 1316

33.12.2 Hiding individual points during machining

In the **FADE** column of the point table, you can specify if the defined point will be hidden during the machining process.

To hide points:

- Select the desired point in the table
- Select the **FADE** column
- Activate **Edit**
- Enter **Y**
- > The control hides the point at the cycle call.

If you enter **Y** in the **FADE** column, you can use the **Skip /** switch to skip this point in **Program Run** operating mode.

Further information: "Icons and buttons", Page 1780

33.13 Datum table

Application

A datum table saves positions on the workpiece. To use a datum table, you must activate it. Within an NC program, the datums can be called, e. g. to execute machining processes on several workpieces at the same position. The active row of the preset table serves as the workpiece preset in the NC program.

Related topics

- Contents and preparing a datum table
  Further information: "Datum table", Page 1858
- Editing a datum table during a program run
  Further information: "Compensation during program run", Page 1795
- Preset table
  Further information: "Preset table", Page 1851

Description of function

Parameters in datum tables

A datum table provides the following parameters:

Parameter	Meaning
D	Row number in the datum table Input: **0...99999999**
X	X coordinate of the datum Input: **-99999.99999...+99999.99999**
Y	Y coordinate of the datum Input: **-99999.99999...+99999.99999**
Z	Z coordinate of the datum Input: **-99999.99999...+99999.99999**
A	A coordinate of the datum Input: **-360.0000000...+360.0000000**
B	B coordinate of the datum Input: **-360.0000000...+360.0000000**
C	C coordinate of the datum Input: **-360.0000000...+360.0000000**
U	U coordinate of the datum Input: **-99999.99999...+99999.99999**
V	V coordinate of the datum Input: **-99999.99999...+99999.99999**
W	W coordinate of the datum Input: **-99999.99999...+99999.99999**
DOC	**Comment on shift?** Input: **Text width 15**

33.13.1 Creating a datum table

To create a datum table:

- Select the **Tables** operating mode
- Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- Select **Create new table**
- > The control opens the **Create new table** window.
- Select the **d** folder
- Select the desired **prototype**
- > The control opens the **Save as** window.
- Select a folder
- Enter a name
- Select **Create**
- Select **OK**
- > The control opens the datum table.

> The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.
> **Further information:** "Table access with SQL statements", Page 1316

33.13.2 Editing a datum table

You can edit the active datum table during program run.

Further information: "Compensation during program run", Page 1795

To edit a datum table:

- Activate **Edit**
- Select the value
- Edit the value
- Save the edited value, e.g. by selecting a different row

NOTICE

Danger of collision!

The control does not consider the changes made to a datum table or compensation table until the values have been saved. You need to activate the datum or compensation value in the NC program again; otherwise, the control will continue using the previous values.

- Make sure to confirm any changes made to the table immediately, e.g., by pressing the **ENT** key
- Activate the datum or compensation value in the NC program again
- Carefully test the NC program after changing the table values

33.14 Tables for cutting data calculation

Application

The following tables allow you to calculate the cutting data of a tool in the cutting data calculator:

- Table for workpiece materials **WMAT.tab**
  Further information: "Table for workpiece materials WMAT.tab", Page 1861
- Table for tool materials **TMAT.tab**
  Further information: "Table for tool materials TMAT.tab", Page 1861
- Cutting data table ***.cut**
  Further information: "Cutting data table *.cut", Page 1862
- Diameter-dependent cutting data table ***.cutd**
  Further information: "Diameter-dependent cutting data table *.cutd", Page 1863

Related topics

- Cutting data calculator
  Further information: "Cutting data calculator", Page 1397
- Tool management
  Further information: "Tool management ", Page 270

Description of function

Table for workpiece materials WMAT.tab

In the table for workpiece materials **WMAT.tab**, you define the workpiece material. You must save this table in the **TNC:\table** folder.

The table for workpiece materials **WMAT.tab** provides the following parameters:

Parameter	Meaning
WMAT	Workpiece material (e.g., aluminum) Input: **Text width 32**
MAT_CLASS	Material class Categorize the materials into material classes with the same cutting conditions, e.g., in accordance with DIN EN 10027-2. Input: **Text width 32**

Table for tool materials TMAT.tab

In the table for tool materials **TMAT.tab**, you define the tool material. You must save this table in the **TNC:\table** folder.

The table for tool materials **TMAT.tab** provides the following parameters:

Parameter	Meaning
TMAT	Tool material (e.g., solid carbide) Input: **Text width 32**
ALIAS1	Additional designation Input: **Text width 32**
ALIAS2	Additional designation Input: **Text width 32**

Cutting data table *.cut

In the cutting data table ***.cut**, you assign the matching cutting data to the workpiece materials and the tool materials. You must save the table in the **TNC:\system\Cutting-Data** folder.

The cutting data table ***.cut** provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows Input: **0...999999999**
MAT_CLASS	Workpiece material from the table **WMAT.tab** **Further information:** "Table for workpiece materials WMAT.tab", Page 1861 Selection by means of a selection window Input: **0...9999999**
MODE	Machining mode (e.g., roughing or finishing) Input: **Text width 32**
TMAT	Tool material from the table **TMAT.tab** **Further information:** "Table for tool materials TMAT.tab", Page 1861 Selection by means of a selection window Input: **Text width 32**
VC	Cutting speed in m/min **Further information:** "Cutting data", Page 280 Input: **0...1000**
FTYPE	Type of feed: ■ **FU**: Feed per revolution **FU** in mm/rev ■ **FZ**: Feed per tooth **FZ** in mm/tooth **Further information:** "Feed rate F", Page 282 Input: **FU**, **FZ**
F	Feed rate value Input: **0.0000...9.9999**

Diameter-dependent cutting data table *.cutd

In the diameter-dependent cutting data table ***.cutd**, you assign the matching cutting data to the workpiece materials and the tool materials. You must save the table in the **TNC:\system\Cutting-Data** folder.

The diameter-dependent cutting data table ***.cutd** provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows Input: **0...999999999**
MAT_CLASS	Workpiece material from the table **WMAT.tab** **Further information:** "Table for workpiece materials WMAT.tab", Page 1861 Selection by means of a selection window Input: **0...9999999**
MODE	Machining mode (e.g., roughing or finishing) Input: **Text width 32**
TMAT	Tool material from the table **TMAT.tab** **Further information:** "Table for tool materials TMAT.tab", Page 1861 Selection by means of a selection window Input: **Text width 32**
VC	Cutting speed in m/min **Further information:** "Cutting data", Page 280 Input: **0...1000**
FTYPE	Type of feed: ■ **FU**: Feed per revolution **FU** in mm/rev ■ **FZ**: Feed per tooth **FZ** in mm/tooth **Further information:** "Feed rate F", Page 282 Input: **FU**, **FZ**
F_D_0...**F_D_9999**	Feed rate value for the respective diameter You don't need to define all columns. If a tool diameter is between two defined columns, the control linearly interpolates the feed rate. Input: **0.0000...9.9999**

Note

In the corresponding folders, the control provides sample tables for automatic cutting data calculation. You can customize theses tables and specify your own data, i.e. materials and tools to be used.

33.15 Pallet table

Application

Pallet tables allow you to define the sequence in which the control will machine the pallets and the NC programs to be used.

Without a pallet changer, you can use pallet tables to successively run NC programs with different presets with just one press of **NC Start**. This type of usage is also called job list.

Tool-oriented machining is possible with pallet tables and with job lists. The control will reduce the number of tool changes, thereby reducing the machining time.

Related topics

- Editing and executing a pallet table in the **Job list** workspace
  Further information: "Job list workspace", Page 1764
- Tool-oriented machining
  Further information: "Tool-oriented machining", Page 1772

Requirement

- Software option 22: Pallet Management

Description of function

Pallet tables can be opened in **Tables**, **Editor**, and **Program Run** operating modes. In **Editor** and **Program Run** operating modes, the control opens the pallet table in the **Job list** workspace instead of as a table.

The machine manufacturer defines a prototype for the pallet table. When you create a new pallet table, the control will copy this prototype. This means that the pallet table on your control might not contain all possible parameters.

The prototype can include the following parameters:

Parameter	Meaning
NR	Row number in the pallet table The entry is required for the **Line number** input field of the **BLOCK SCAN** function. **Further information:** "Block scan for mid-program startup", Page 1786 Input: **0...99999999**
TYPE	**Pallet type?** Contents of the table row: ■ **PAL**: Pallet ■ **FIX**: Fixture ■ **PGM**: NC program Selection using a selection menu Input: **PAL**, **FIX**, **PGM**
NAME	**Pallet / NC program / Fixture?** File name of the pallet, fixture or NC program The machine manufacturer specifies the names of pallets and fixtures as needed. You can define the names of your NC programs yourself. Selection by means of a selection window Input: **Text width 32**

Parameter	Meaning
DATUM	**Datum table?** The datum table to be used in the NC program. Selection by means of a selection window Input: **Text width 32**
PRESET	**Preset?** Row number in the preset table for the workpiece preset to be activated. Selection by means of a selection window Input: **0...999**
LOCATION	**Location?** The entry **MA** indicates that there is a pallet or fixture in the working space of the machine and can be machined. Press the **ENT** key to enter **MA**. Press the **NO ENT** key to remove the entry and thus suppress machining. If the column exists, an entry is mandatory. Corresponds to the **Machinable** switch in the **Form** workspace. Selection using a selection menu Input: No value, **MA**
LOCK	**Locked?** By entering the character *****, you can exclude the row of the pallet table from machining. Press the **ENT** key to enter ***** for the row. Press the **NO ENT** key to cancel the lock. You can lock the execution for individual NC programs, fixtures or entire pallets. Unlocked rows (e.g., PGM) in a locked pallet are also not machined. Selection using a selection menu Input: No value, *****
W STATUS	**Machining status?** Relevant to tool-oriented machining The machining status defines the machining progress. Enter BLANK for an unmachined (raw) workpiece. The control changes this entry automatically during machining. The control differentiates between the following entries ■ BLANK / no entry: Workpiece blank, requires machining ■ INCOMPLETE: Partly machined, requires further machining ■ ENDED: Machined completely, no further machining required ■ EMPTY: Empty space, no machining required ■ SKIP: Skip machining **Further information:** "Tool-oriented machining", Page 1772 Input: No value, **BLANK**, **INCOMPLETE**, **ENDED**, **EMPTY**, **SKIP**
PALPRES	**Pallet preset** Row number in the pallet preset table for the pallet preset to be activated Only required if a pallet preset table has been created on the control. Selection by means of a selection window Input: **-1...+999**
DOC	Comment Input: **Text width 15**

Parameter	Meaning
METHOD	**Machining method?** Machining method The control differentiates between the following entries ▪ WPO: Workpiece oriented (standard) ▪ TO: Tool oriented (first workpiece) ▪ CTO: Tool oriented (further workpieces) **Further information:** "Tool-oriented machining", Page 1772 Selection using a selection menu Input: **WPO**, **TO**, **CTO**
CTID	**ID no. geometry context?** Relevant to tool-oriented machining The control automatically generates the ID number for mid-program startup with block scan. If you delete or change the entry, mid-program startup is no longer possible. **Further information:** "Tool-oriented machining", Page 1772 Input: **Text width 8**
SP-X	**Clearance height?** Clearance height in the X axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1772 Input: **-999999.99999...+999999.99999**
SP-Y	**Clearance height?** Clearance height in the Y axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1772 Input: **-999999.99999...+999999.99999**
SP-Z	**Clearance height?** Clearance height in the Z axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1772 Input: **-999999.99999...+999999.99999**
SP-A	**Clearance height?** Clearance height in the A axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1772 Input: **-999999.99999...+999999.99999**
SP-B	**Clearance height?** Clearance height in the B axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1772 Input: **-999999.99999...+999999.99999**
SP-C	**Clearance height?** Clearance height in the C axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1772 Input: **-999999.99999...+999999.99999**
SP-U	**Clearance height?** Clearance height in the U axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1772 Input: **-999999.99999...+999999.99999**

Parameter	Meaning
SP-V	**Clearance height?** Clearance height in the V axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1772 Input: **-999999.99999...+999999.99999**
SP-W	**Clearance height?** Clearance height in the W axis for tool-oriented machining **Further information:** "Tool-oriented machining", Page 1772 Input: **-999999.99999...+999999.99999**
COUNT	**Number of operations** For rows of the **PAL** type: Current actual value for the pallet counter nominal value defined in the **TARGET** column. For rows of the **PGM** type: Value indicating by how much the pallet counter actual value will be incremented after the execution of the NC program. **Further information:** "Pallet counter", Page 1764 Input: **0...99999**
TARGET	**Total number of operations** Nominal value for the pallet counter in rows of the **PAL** type The control repeats the NC programs of this pallet until the nominal value has been reached. **Further information:** "Pallet counter", Page 1764 Input: **0...99999**

33.15.1 Creating and opening a pallet table

To create a pallet table:

- Select the **Tables** operating mode
- Select **Add**
- The control opens the **Quick selection** and the **Open File** workspaces.
- Select **Create new table**
- The control opens the **Create new table** window.
- Select the **p** folder
- Select table format
- Select a path: Select **Select a path**
- The control opens the **Save as** window.
- Select a folder
- Enter a name
- Create: Select **Create**
- The control opens the table in **Tables** operating mode.

- The file name of a pallet table must always begin with a letter.
- Use the **Select in Program Run** button in **Files** operating mode to open the pallet table in **Program Run** operating mode. In this operating mode, you can edit and execute pallet tables.

  Further information: "Job list workspace", Page 1764

33.16 Compensation tables

33.16.1 Overview

The control provides the following compensation tables:

Table	Further information
Compensation table ***.tco** Compensation in the tool coordinate system **T-CS**	Page 1868
Compensation table ***.wco** Compensation in the working plane coordinate system **WPL-CS**	Page 1870

33.16.2 Compensation table *.tco

Application

The compensation table ***.tco** allows you to define compensation values for the tool in the tool coordinate system **T-CS**.

You can use the compensation table ***.tco** for tools of all types of technologies.

Related topics

- Using compensation tables
  Further information: "Tool compensation with compensation tables", Page 1044
- Contents of the compensation table ***.wco**
  Further information: "Compensation table *.wco", Page 1870
- Editing compensation tables during program run
  Further information: "Compensation during program run", Page 1795
- Tool coordinate system **T-CS**
  Further information: "Tool coordinate system T-CS", Page 946

Description of function

Any compensation in the compensation tables with the ***.tco** file name extension applies to the active tool. The table applies to all tool types. Therefore, columns that you may not need for your specific tool type will be displayed during creation.

Enter only those values that are relevant to your tool. If you compensate for values that are not present with the existing tool, the control issues an error message.

The compensation table ***.tco** provides the following parameters:

Parameter	Meaning
NO	Row number in the table Input: **0...999999999**
DOC	Comment Input: **Text width 16**
DL	**Tool length oversize?** Delta value for parameter **L** of the tool table Input: **-999.9999...+999.9999**
DR	**Tool radius oversize?** Delta value for parameter **R** of the tool table Input: **-999.9999...+999.9999**
DR2	**Tool radius oversize 2?** Delta value for parameter **R2** of the tool table Input: **-999.9999...+999.9999**
DXL	**Oversize in tool length 2?** Delta value for parameter **DXL** of the turning tool table Input: **-999.9999...+999.9999**
DYL	**Tool length oversize 3?** Delta value for parameter **DYL** of the turning tool table Input: **-999.9999...+999.9999**
DZL	**Oversize in tool length 1?** Delta value for parameter **DZL** of the turning tool table Input: **-999.9999...+999.9999**
DL-OVR	**Compensation of the overhang** Delta value for parameter **L-OVR** of the grinding tool table Input: **-999.9999...+999.9999**
DR-OVR	**Compensation of the radius** Delta value for parameter **R-OVR** of the grinding tool table Input: **-999.9999...+999.9999**
DLO	**Compensation of the total length** Delta value for parameter **LO** of the grinding tool table Input: **-999.9999...+999.9999**
DLI	**Compensation of the length to the inner edge** Delta value for parameter **LI** of the grinding tool table Input: **-999.9999...+999.9999**

33.16.3 Compensation table *.wco

Application

The values from the compensation tables with the ***.wco** file name extension are applied as shifts in the working plane coordinate system (**WPL-CS**).

The ***.wco** compensation tables are used mainly for turning (option 50).

Related topics

- Using compensation tables
  Further information: "Tool compensation with compensation tables", Page 1044
- Contents of the compensation table ***.tco**
  Further information: "Compensation table *.tco", Page 1868
- Editing compensation tables during program run
  Further information: "Compensation during program run", Page 1795
- Working plane coordinate system **WPL-CS**
  Further information: "Working plane coordinate system WPL-CS", Page 942

Description of function

The compensation table ***.wco** provides the following parameters:

Parameter	Meaning
NO	Row number in the table Input: **0...999999999**
DOC	Comment Input: **Text width 16**
X	Shift of the working plane coordinate system **WPL-CS** in **X** Input: **-999.9999...+999.9999**
Y	Shift of **WPL-CS** in **Y** Input: **-999.9999...+999.9999**
Z	Shift of **WPL-CS** in **Z** Input: **-999.9999...+999.9999**

33.16.4 Creating a compensation table

To create a compensation table:

- ▶ Select the **Tables** operating mode
- ▶ Select **Add**
- \> The control opens the **Quick selection** and the **Open File** workspaces.
- ▶ Select **Create new table**
- \> The control opens the **Create new table** window.
- ▶ Select the folder **tco** or **wco**

Select a path

- ▶ Select **Select a path**
- \> The control opens the **Save as** window.
- ▶ Select a folder
- ▶ Enter a name

Create

- ▶ Select **Create**
- \> The control opens the table.

33.17 *.3DTC compensation table

Application

In a ***.3DTC** compensation table, the control saves the radius deviation of ball-nose cutters from the nominal value at a defined inclination angle. For workpiece touch probes, the control saves the deflection behavior of the touch probe at a defined probing angle.

The control takes into account the saved data during the execution of NC programs and during probing.

Related topics

- 3D radius compensation depending on the tool's contact angle
  Further information: "3D radius compensation depending on the tool contact angle (option 92)", Page 1062
- 3D calibration of the touch probe
  Further information: "Calibrating the workpiece touch probe", Page 1439

Requirements

- Advanced Functions Set 2 (software option 9)
- 3D-ToolComp (software option 92)

Description of function

The ***.3DTC** compensation tables must be saved in the **TNC:\system\3D-ToolComp** folder. In the **DR2TABLE** tool management column, you can then assign the tables to a tool.

You create a separate table for each tool.

A compensation table provides the following parameters:

Parameter	Meaning
NR	Sequential row number in the compensation table The control evaluates a maximum of 100 rows in the compensation value table. Input: **0...9999999**
ANGLE	Inclination angle of tools or probing angle of workpiece touch probes Input: **-99999.999999...+99999.999999**
DR2	Radius deviation from the nominal value or deflection of the touch probe Input: **-99999.999999...+99999.999999**

33.18 Tables for AFC (option 45)

33.18.1 Basic AFC settings in AFC.tab

Application

In the **AFC.TAB** table, you can enter the feed rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.

Related topics

- Programming AFC
  Further information: "Adaptive Feed Control (AFC, option 45)", Page 1114

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

The data in this table are default values that are copied into a file belonging to the respective NC program during a teach-in cut. The values act as the basis for feedback control.

Further information: "Description of function", Page 1876

> If you define a tool-specific feedback-control reference power using the **AFC-LOAD** column in the tool table, the control generates the associated file for the relevant NC program without a teach-in cut. The file is created shortly before feedback control becomes effective.

Parameter

The **AFC.tab** table provides the following parameters:

Parameter	Meaning
NR	Row number in the table Input: **0...9999**
AFC	Name of the control setting Enter this name in the **AFC** tool management column. It specifies the assignment of the control parameters to the tool. Input: Text width 10
FMIN	Feed rate at which the control will perform an overload reaction Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. **Further information:** "Monitoring tool wear and tool load", Page 1120 Input: **0...999**
FMAX	Maximum feed rate within the material up to which the control can automatically increase the feed rate Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. **Further information:** "Monitoring tool wear and tool load", Page 1120 Input: **0...999**
FIDL	Feed rate at which the control will traverse the tool outside of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) Input: **0...999**
FENT	Feed rate at which the control will move the tool into and out of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) Input: **0...999**

<table>
<tr><th>Parameter</th><th>Meaning</th></tr>
<tr><td>OVLD</td><td>Desired reaction of the control to overload:
M: Execution of a macro defined by the machine manufacturer
S: Immediate NC stop
F: NC stop once the tool has been retracted
E: Just display an error message on the screen
L: Disable active tool
-: No overload reaction
If the maximum spindle power is exceeded for more than one second and the feed rate falls below the defined minimum during that time, the control will conduct an overload reaction.
In conjunction with the cut-related tool wear monitoring function, the control will only evaluate the options M, E, and L!
Input: M, S, F, E, L, or -</td></tr>
<tr><td>POUT</td><td>Spindle power at which the control will detect that the tool exits the workpiece
Enter the value in percent of the learned reference load
Recommended input value: 8%
In turning mode: Minimum load Pmin for tool monitoring (option 50)
Input: 0...100</td></tr>
<tr><td>SENS</td><td>Sensitivity (aggressiveness) of feedback control
50 is for slow feedback control, 200 for a very aggressive feedback control. An aggressive feedback control responds quickly and significantly changes the values, but it tends to overshoot.
In turning mode: Activate the monitoring of the minimum load Pmin (option 50):
1: Evaluate Pmin
0: Do not evaluate Pmin
Input: 0...999</td></tr>
<tr><td>PLC</td><td>Value that the control will transfer to the PLC at the beginning of a machining step
The machine manufacturer defines whether and which function will be performed by the control.
Input: 0...999</td></tr>
</table>

Creating an AFC.tab table

You need to create the table only if the table is missing in the **tables** folder.

To create the **AFC.tab** table:

- ▶ Select the **Tables** operating mode
- ▶ Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- ▶ Select **Create new table**
- > The control opens the **Create new table** window.
- ▶ Select the **tab** folder
- ▶ Select the **AFC.TAB** format
- ▶ Select **Select a path**
- > The control opens the **Save as** window.
- ▶ Select a folder
- ▶ Enter a name
- ▶ Select **Create**
- > The control opens the table.

Notes

- If there is no AFC.TAB table in the **TNC:\table** directory, the control uses a permanently defined, internal control setting for the teach-in cut. If, alternatively, a tool-dependent reference power value exists, the control uses it immediately. HEIDENHAIN recommends using the AFC.TAB table in order to ensure safe and well-defined operation.
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.

  Further information: "Table access with SQL statements", Page 1316

33.18.2 AFC.DEP settings file for teach-in cuts

Application

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. **<name>** is the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value to the table.

Related topics

- AFC basic settings in the table **AFC.tab**

  Further information: "Basic AFC settings in AFC.tab", Page 1872
- Setting up and using AFC

  "Adaptive Feed Control (AFC, option 45)"

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

Each row in the **<name>.H.AFC.DEP** file stands for a machining section, that you start with **FUNCTION AFC CUT BEGIN** and complete with **FUNCTION AFC CUT END**. You can edit all data of the **<name>.H.AFC.DEP** file for optimization purposes. If you have optimized the values from the AFC.TAB table, the control places a ***** in front of these control settings in the AFC column.

Further information: "Basic AFC settings in AFC.tab", Page 1872

In addition to the contents from the **AFC.tab** table, the **AFC.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed (not editable)
IDX	Index of the tool with which the machining step was performed (not editable)
N	Difference for tool call: ■ **0**: Tool was called by its tool number ■ **1**: Tool was called by its tool name
PREF	Reference load of the spindle. The control measures the value in percent with respect to the rated spindle power
ST	Status of the machining step: ■ **L**: In the next program run, a teach-in cut will be recorded for this machining step. The control will overwrite any existing values in this line ■ **C**: The teach-in cut was completed successfully. The next program run can be conducted with automatic feed control
AFC	Name of the control setting

Notes

- Note that the **<name>.H.AFC.DEP** file is locked against editing as long as the NC program **<name>.H** is running.

  The control does not remove the editing lock until one of the following functions has been executed:
  - **M2**
  - **M30**
  - **END PGM**
- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control will display the dependent files in the file manager.

33.18.3 Log file AFC2.DEP

Application

The control stores various pieces of information for each machining step of a teach-in cut in the **<name>.H.AFC2.DEP** file. **<name>** is the name of the NC program for which you have recorded the teach-in cut. During feedback control, the control updates the data and performs various evaluations.

Related topics

- Setting up and using AFC
  Further information: "Adaptive Feed Control (AFC, option 45)", Page 1114

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

The **AFC2.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed
IDX	Index of the tool with which the machining step was performed
SNOM	Nominal spindle speed [rpm]
SDIFF	Maximum difference of the spindle speed in % of the nominal speed
CTIME	Machining time (tool in effect)
FAVG	Average feed rate (tool in effect)
FMIN	Smallest occurring feed factor. The control shows the value as a percentage of the programmed feed rate
PMAX	Maximum recorded spindle power during machining. The control shows the value as a percentage of the spindle's rated power.
PREF	Reference load of the spindle. The control shows the value as a percentage of the spindle's rated power.
OVLD	Overload reaction performed by the control: ■ **M**: A macro defined by the machine tool builder has been run ■ **S:** Immediate NC stop was conducted ■ **F**: NC stop was conducted after the tool was retracted ■ **E**: An error message was displayed ■ **L**: The current tool was locked ■ **-**: There was no overload reaction
BLOCK	Block number at which the machining step begins

During feedback control, the control determines the current machining time as well as the resulting time saving in percent. The control enters the results of the evaluation between the key words **total** and **saved** in the last line of the log file. Where the time balance is positive, the percentage value is also positive.

Note

- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control will display the dependent files in the file manager.

33.18.4 Editing tables for AFC

You can open and, if necessary, edit the tables for AFC during program run. The control offers only the tables for the active NC program.

To open a table for AFC:

- Select the **Program Run** operating mode

AFC settings

- Select **AFC settings**
- > The control displays a selection menu. The control shows all existing tables for this NC program.
- Select a file, e.g. **AFC.TAB**
- > The control opens the file in the **Tables** operating mode.

33.19 Technology table for Cycle 287 Gear Skiving

Application

In Cycle **287 GEAR SKIVING**, you can use the cycle parameter **QS240 NUMBER OF CUTS** to call a table containing technology data. The table is a freely definable table, and as such is in ***.tab** format. The control provides you with a template. In the table, you define the following data for each individual cut:

- Feed rate
- Lateral infeed
- Lateral offset

Requirements

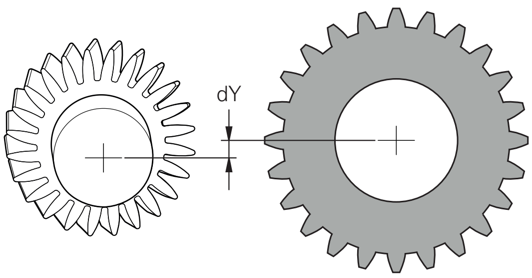
- Software option 157: Gear Cutting

33.19.1 Parameters in the technology table

Parameter in the table

The technology data table contains the following parameters:

Parameter	Function
NR	Number of the cut that also corresponds to the number of the table row
FEED	Feed rate in mm/rev or 1/10 inch/rev for the cut This parameter replaces the following cycle parameters: ■ **Q588 FIRST FEED RATE** ■ **Q589 LAST FEED RATE** ■ **Q580 FEED-RATE ADAPTION** Input: **0...9999.999**
INFEED	Lateral infeed of the cut. This entry is incremental. This parameter replaces the following cycle parameters: ■ **Q586 FIRST INFEED** ■ **Q587 LAST INFEED** Input: **0...99.99999**
dY	Lateral offset of the cut (to improve the removal of chips). Input: **-9.99999...+9.99999**



Notes

- The unit used in the NC program determines whether millimeter or inch units are used.
- In order to avoid contour distortions, HEIDENHAIN recommends that you do not program an offset **dY** in the last cut.
- HEIDENHAIN recommends that you program only minimum offset values **dY** in the individual cuts, because this might result in contour damage.
- The sum of the lateral infeeds (**INFEED**) must result in the tooth height.
  - If the tooth height is greater than the total infeed, the control will display a warning.
  - If the tooth height is less than the total infeed, the control will display an error message.

Example:

- **TOOTH HEIGHT** (**Q563**) = 2 mm
- Number of cuts (**NR**) = 15
- Lateral infeed (**INFEED**) = 0.2 mm
- Total infeed = **NR** * **INFEED** = 3 mm

  In this case, the tooth height is less than the total infeed (2 mm < 3 mm). Reduce the number of cuts to 10.

33.19.2 Creating a technology table

To create a table containing technology data:

- ▶ Select the **Tables** operating mode
- ▶ Select **Add**
- > The control opens the **Quick selection** and the **Open File** workspaces.
- ▶ Select **Create new table**
- > The control opens the **Create new table** window.
- ▶ Select the **tab** folder
- ▶ Select the **Proto_Skiving.TAB** format
- ▶ Select **Select a path**
- > The control opens the **Save as** window.
- ▶ Select a folder
- ▶ Enter name
- ▶ Select **Create**
- > The control opens the technology table.

34

Electronic Handwheel

34.1 Fundamentals

Application

If you want to approach a position in the machine's working space while the guard door is open or if you execute a small infeed movement, you can use the electronic handwheel. The electronic handwheel allows you to traverse the axes and perform various functions provided by the control.

Related topics

- Incremental jog positioning

  Further information: "Incremental jog positioning of axes", Page 183
- Handwheel superimpositioning with GPS (option 44)

  Further information: "Function Handwheel superimp.", Page 1141
- Handwheel superimpositioning with **M118**

  Further information: "Activating handwheel superimpositioning with M118", Page 1239
- Virtual tool axis **VT**

  Further information: "Virtual tool axis VT", Page 1142
- Touch probe functions in **Manual** operating mode

  Further information: "Touch Probe Functions in the Manual Operating Mode", Page 1425

Requirement

- Electronic handwheel (e.g., HR 550FS)

  The control supports the following electronic handwheels:
  - HR 410: Cable-bound handwheel without display
  - HR 420: Cable-bound handwheel with display
  - HR 510: Cable-bound handwheel without display
  - HR 520: Cable-bound handwheel with display
  - HR 550FS: Wireless handwheel with display, data transmission via radio

Description of function

You can use electronic handwheels in **Manual** or **Program Run** operating mode.

The HR 520 and HR 550FS portable handwheels feature a display that allows the control to show different types of information. You can use the handwheel soft keys for setup functions, such as the setting of presets or the activation of miscellaneous functions.

Once you have activated the handwheel with the handwheel activation key or the **Handwheel** switch, you can operate the control only by using the handwheel. If you press the axis keys in this state, the control will display the message **Handwheel active: Handwheel-1, MB0**.

If more than one handwheel is connected to a control, you can activate or deactivate a handwheel only by pressing the handwheel activation key on the respective handwheel. You need to deactivate the active handwheel in order to be able to select another handwheel.

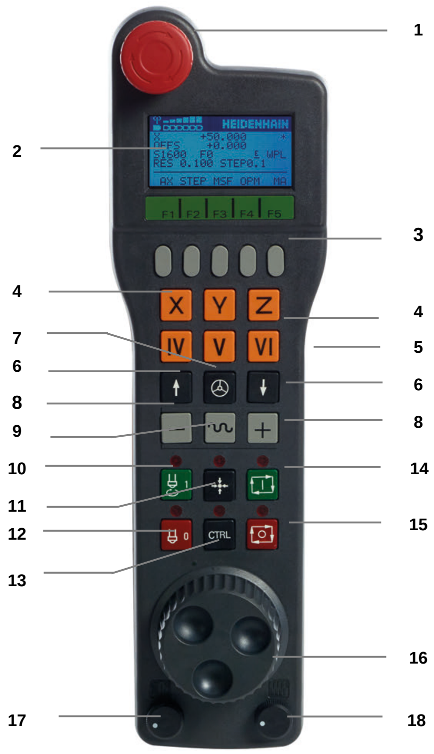
Functions in Program Run operating mode

You can perform the following functions in **Program Run** operating mode:

- The **NC Start** key (**NC Start** handwheel key)
- The **NC Stop** key (**NC Stop** handwheel key)
- After the **NC Stop** key has been pressed: Internal stop (handwheel soft keys **MOP** and then **Stop**)
- After the **NC STOP** key has been pressed: Traverse manual axes (handwheel soft keys **MOP** and then **MAN**)
- Return to the contour after axes were manually traversed during an interruption of the program run (handwheel soft keys **MOP** and then **REPO**). The handwheel soft keys are used for operating.

  Further information: "Returning to the contour", Page 1793
- Switch on/off the "Tilt working plane" function (handwheel soft keys **MOP** and then **3-D**)

Operating elements of an electronic handwheel



An electronic handwheel provides the following operating elements:

1 **EMERGENCY STOP** key
2 Handwheel display for status and for selecting functions
3 Handwheel soft keys
4 Axis keys; can be exchanged by the machine manufacturer depending on the axis configuration
5 Permissive button

The permissive button is on the rear side of the handwheel.

6 Arrow keys for defining the handwheel resolution
7 Handwheel activation key

8 Axis-direction key

Key for the direction of the traverse motion

9 Rapid traverse override for the traverse motion

10 Spindle switch-on (machine-dependent function, key can be exchanged by the machine manufacturer)

11 **Generate NC block** key (machine-dependent function, key can be exchanged by the machine manufacturer)

12 Spindle switch-off (machine-dependent function, key can be exchanged by the machine manufacturer)

13 **CTRL** key for special functions (machine-dependent function, key can be exchanged by the machine manufacturer)

14 **NC START** key (machine-dependent function, key can be exchanged by the machine manufacturer)

15 **NC Stop** key

Machine-dependent function; key can be exchanged by the machine manufacturer

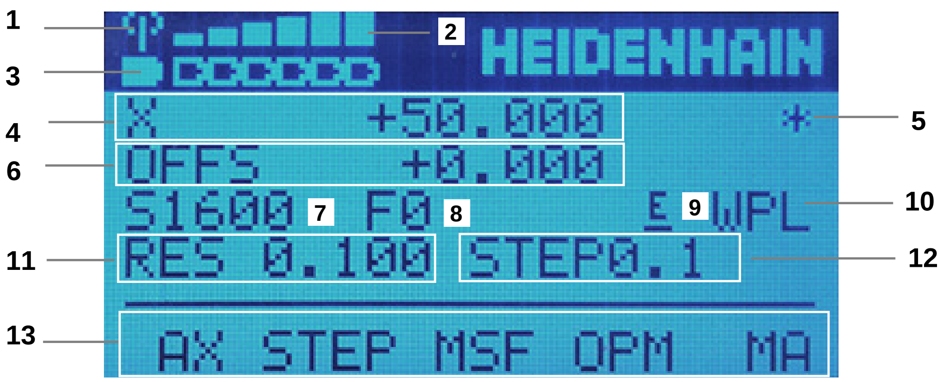
16 Handwheel

17 Spindle speed potentiometer

18 Feed rate potentiometer

19 Cable connection; not required for the HR 550 FS wireless handwheel

Contents of an electronic handwheel display



The display of an electronic handwheel consists of the following areas:

1 Handwheel is in the docking station or radio mode is active

Only with HR 550 FS wireless handwheel

2 Field strength

Six bars = maximum field strength

Only with HR 550 FS wireless handwheel

3 Charging condition of battery

Six bars = maximum charging condition. A bar moves from the left to the right during recharging.

Only with HR 550 FS wireless handwheel

4 **X+50.000**: Position of the selected axis

5 *****: STIB (control in operation); program run has been started or axis is in motion

6 Handwheel superimpositioning from **M118** or the Global Program Settings GPS (option 44)

  Further information: "Activating handwheel superimpositioning with M118", Page 1239

  Further information: "Function Handwheel superimp.", Page 1141

7 **S1000**: Current spindle speed

8 Feed rate at which the selected axis is moving

9 **E**: Error message

  If an error message appears on the control, the handwheel display shows the message **ERROR** for three seconds. Then the letter **E** is shown in the display as long as the error is pending on the control.

10 Active setting in the **3-D rotation** window:

  - **VT**: **Tool axis** function
  - **WP**: **Basic rotation** function
  - **WPL**: **3D ROT** function

  Further information: "3-D rotation window (option 8)", Page 1022

11 Handwheel resolution

  Distance that the selected axis moves per handwheel revolution

  Further information: "Handwheel resolution", Page 1886

12 Incremental jog active or inactive

  If the function is active, the control will display the active traverse step.

13 Soft-key row

  The soft key row provides the following functions:

  - **AX**: Select the machine axis

    Further information: "Creating a positioning block", Page 1888
  - **STEP**: Incremental jog positioning

    Further information: "Incremental jog positioning", Page 1888
  - **MSF**: Execute various functions of **Manual** operating mode (e.g., entering the feed rate **F**)

    Further information: "Entering miscellaneous functions M", Page 1887
  - **OPM**: Select the operating mode
    - **MAN**: **Manual** operating mode
    - **MDI**: **MDI** application in **Manual** operating mode
    - **RUN**: **Program Run** operating mode
    - **SGL**: **Single Block** mode of **Program Run** operating mode
  - **MA**: Switching the magazine pockets

Handwheel resolution

The handwheel sensitivity determines which path an axis takes per revolution of the handwheel. The handwheel sensitivity results from the defined handwheel speed of the axis and the speed level used internally by the control. The speed level describes a percentage of the handwheel speed. The control calculates a specific handwheel sensitivity value for each speed level. The resulting handwheel sensitivity values are directly selectable with the handwheel arrow keys (only if incremental jog is not active).

The handwheel speed indicates the increment (e.g., 0.01 mm) traversed per handwheel detent position. You can change the handwheel speed with the handwheel's arrow keys.

If you have defined a handwheel speed of 1, the following handwheel resolutions are available:

Resulting handwheel sensitivity levels in mm/revolution and degrees/revolution:
0.0001/0.0002/0.0005/0.001/0.002/0.005/0.01/0.02/0.05/0.1/0.2/0.5/1

Resulting handwheel sensitivity levels in in/revolution:
0.000127/0.000254/0.000508/0.00127/0.00254/0.00508/0.0127/0.0254/0.0508/0.127/0.254/0.508

Examples for resulting handwheel sensitivity values:

Defined handwheel speed	Speed level	Resulting handwheel sensitivity
10	0.01 %	0.001 mm/revolution
10	0.01 %	0.001 degrees/revolution
10	0.0127 %	0.00005 in/revolution

Effect of the feed-rate potentiometer when handwheel is active

NOTICE

Caution: Possible damage to the workpiece!

When toggling between the machine operating panel and the handwheel, the feed rate may be reduced. This can cause visible marks on the workpiece.

- Make sure to retract the tool before toggling between the handwheel and the machine operating panel.

The settings of the feed-rate potentiometer on the handwheel may differ from those on the machine operating panel. When you activate the handwheel, the control will automatically activate the feed-rate potentiometer of the handwheel. When you deactivate the handwheel, the control will automatically activate the feed-rate potentiometer of the machine operating panel.

In order to make sure that the feed rate does not increase while you switch over between the potentiometers, the feed rate is either frozen or reduced.

If the feed rate before switching over is higher than the feed rate after switching over, the control automatically reduces the feed rate to the smaller value.

If the feed rate before switching over is less than the feed rate after switching over, the control automatically freezes the feed rate. In this case, you must turn the feed-rate potentiometer back to the previous value because the activated feed-rate potentiometer will only then be effective.

34.1.1 Entering spindle speed S

To enter the spindle speed **S** by using an electronic handwheel:

- Press the **F3** (**MSF**) handwheel soft key
- Press the **F2** (**S**) handwheel soft key
- Select the desired speed by pressing the **F1** or **F2** key
- Press the **NC Start** key
- > The control activates the entered spindle speed.

> If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.
>
> By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

34.1.2 Entering the feed rate F

To enter the feed rate **F** by using an electronic handwheel:

- Press the **F3** (**MSF**) handwheel soft key
- Press the **F3** (**F**) handwheel soft key
- Select the desired feed rate by pressing the **F1** or **F2** key
- Confirm the new feed rate F with the **F3** (**OK**) handwheel soft key

> If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a decimal value of 0.
>
> By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

34.1.3 Entering miscellaneous functions M

To enter a miscellaneous function by using an electronic handwheel:

- Press the **F3** (**MSF**) handwheel soft key
- Press the **F1** (**M**) handwheel soft key
- Select the desired M function number by pressing the **F1** or **F2** key
- Press the **NC Start** key
- > The control activates the miscellaneous function

Further information: "Overview of miscellaneous functions", Page 1225

34.1.4 Creating a positioning block

Refer to your machine manual.

Your machine tool builder can assign any function to the **Generate NC block** handwheel key.

To create a positioning block by using an electronic handwheel:

- Select the **Manual** operating mode
- Select the **MDI** application
- If necessary, select the NC block after which the positioning block should be inserted
- Activate the handwheel
- Press the **Generate NC block** key on the handwheel
- > The control inserts a straight line **L**, including all of the axis positions.

34.1.5 Incremental jog positioning

Incremental jog positioning allows you to move the selected axis by a preset value.

To incrementally position an axis by using an electronic handwheel:

- Press the handwheel soft key F2 (**STEP**)
- Press the handwheel soft key 3 (**ON**)
- > The control activates incremental jog positioning.
- Set the desired jog increment by using the **F1** or **F2** keys

The smallest possible increment is 0.0001 mm (0.00001 inches). The largest possible increment is 10 mm (0.3937 inches)..

- Confirm the selected jog increment by pressing the handwheel soft key F4 (**OK**)
- Use the **+** or **-** handwheel key to move the active handwheel axis in the corresponding direction
- > The control moves the active axis by the entered increment every time the handwheel key is pressed.

If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10.

By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Notes

⚠ DANGER
Caution: hazard to the user!
Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!
▶ Devices should be connected or removed only by authorized service technicians
▶ Only switch on the machine via a connected handwheel or a secured connection

NOTICE
Caution: Danger to the tool and workpiece!
The wireless handwheel triggers an emergency stop reaction if the radio transmission is interrupted, the battery is fully empty, or if there is a defect. Emergency stop reactions during machining can cause damage to the tool or workpiece.
▶ Place the handwheel in the handwheel holder when it is not in use
▶ Keep the distance between the handwheel and the handwheel holder small (pay attention to the vibration alarm)
▶ Test the handwheel before machining

- The machine manufacturer can provide additional functions for the HR5xx handwheels.

  Refer to your machine manual.
- You can use the axis keys to activate the **X**, **Y**, and **Z** axes, as well as three other axes that can be defined by the machine manufacturer. Your machine manufacturer can also place the virtual axis **VT** on one of the free axis keys.

34.2 HR 550FS wireless handwheel

Application

With the HR 550FS wireless handwheel and its radio transmission characteristics, you can move farther away from the machine operating panel than with other handwheels. The HR 550FS wireless handwheel thus provides an important benefit, in particular for large machines.

Description of function

The HR 550FS wireless handwheel comes fitted with a rechargeable battery. The battery starts charging when you place the handwheel into the holder.

The HRA 551FS handwheel holder and the HR 550FS handwheel together form one function unit.

HR 550FS handwheel

HRA 551FS handwheel holder

The HR 550FS handwheel can be operated by battery for up to eight hours before it needs recharging. A completely discharged handwheel takes approx. three hours for a full charge. When you do not use the HR 550FS, always place it into the handwheel holder. This charges the handwheel battery constantly and a direct connection with the emergency-stop circuit is provided.

When the handwheel is in its holder, it provides the same functionality as during radio mode. This allows you to use a completely discharged handwheel.

> Clean the contacts of the handwheel holder and handwheel regularly to ensure their proper functioning.
>
> If the control has triggered an emergency stop, you must reactivate the handwheel.
>
> **Further information:** "Reactivating the handwheel", Page 1894

If you happen to get close to the limit of the transmission range, the HR 550FS will set off a vibrating alarm. If this occurs, you must reduce the distance to the handwheel holder.

Note

⚠ DANGER

Caution: hazard to the user!

Wireless handwheels, due to their rechargeable batteries and the influence of other wireless devices, are more susceptible to interference than cable-bound connections are. Ignoring the requirements for and information about safe operation leads to endangerment of the user, for example during installation or maintenance work.

- Check the radio connection of the handwheel for possible overlapping with other wireless devices
- Switch off the wireless handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the control can run a functional test when it is restarted
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the handwheel holders (such as with color-coded stickers)
- If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the respective machine (such as with a functional test)

34.3 Configuration of wireless handwheel window

Application

The **Configuration of wireless handwheel** window shows the connection data of the HR 550FS wireless handwheel and provides various functions for optimizing the radio connection, such as setting the radio channel.

Related topics

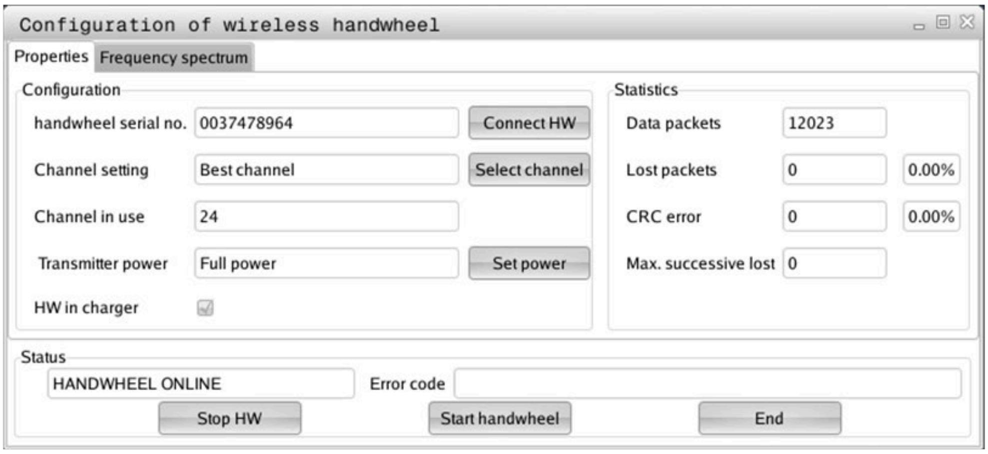
- Electronic handwheel

  Further information: "Electronic Handwheel", Page 1881
- HR 550FS wireless handwheel

  Further information: "HR 550FS wireless handwheel", Page 1889

Description of function

Use the **Set up wireless handwheel** menu item to open the **Configuration of wireless handwheel** window. The menu item is in the **Machine settings** group of the **Settings** application.



Areas of the Configuration of wireless handwheel window

Configuration area

In the **Configuration** area, the control displays different types of information about the connected wireless handwheel, such as the serial number.

Statistics area

In the **Statistics** area, the control displays information about the transmission quality.

If the reception quality is poor so that a proper and safe stop of the axes cannot be ensured anymore, an emergency-stop reaction of the wireless handwheel is triggered.

A high value under **Max. successive lost** is an indication of a limited quality of reception. If the control repeatedly displays values greater than 2 during normal operation of the wireless handwheel within the desired range of use, there is a high risk of undesired disconnection.

If this occurs, try to improve the transmission quality by selecting a different channel or by increasing the transmitter power.

Further information: "Setting the radio channel", Page 1894

Further information: "Selecting the transmission power", Page 1893

Status area

In the **Status** area, the control displays the current status of the handwheel, such as **HANDWHEEL ONLINE** and pending error messages concerning the connected handwheel.

34.3.1 Assigning a handwheel to a handwheel holder

In order to assign a handwheel to a handwheel holder, the handwheel holder must be connected to the control hardware.

To assign a handwheel to a handwheel holder:

- Place the handwheel into the handwheel holder
- Select the **Home** operating mode
- Select the **Settings** application
- Select the **Machine settings** group
- Double-tap or double-click the **Set up wireless handwheel** menu item
- > The control opens the **Configuration of wireless handwheel** window.
- Select the **Connect HW** button
- > The control saves the serial number of the inserted wireless handwheel and displays it in the configuration window to the left of the **Connect HW** button.
- Select the **END** button
- > The control saves the configuration.

34.3.2 Selecting the transmission power

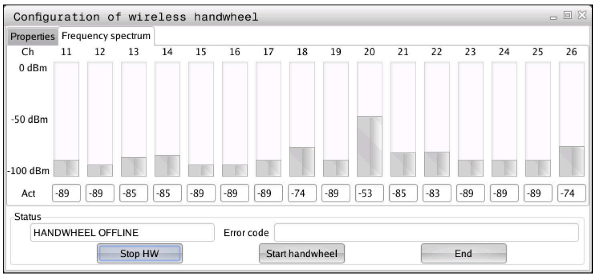
If you reduce the transmission power, the range of the wireless handwheel will decrease.

To set the transmission power of the handwheel:

- Open the **Configuration of wireless handwheel** window
- Select the **Set power** button
- > The control displays the three available power settings.
- Select the desired transmission power setting
- Select the **END** button
- > The control saves the configuration.

34.3.3 Setting the radio channel

If the wireless handwheel is started automatically, then the control tries to select the radio channel providing the best radio signal.



To set the radio channel manually:

- ▶ Open the **Configuration of wireless handwheel** window
- ▶ Select the **Frequency spectrum** tab
- ▶ Select the **Stop HW** button
- > The control stops the connection to the wireless handwheel and determines the current frequency spectrum for all 16 available channels.
- ▶ Note the number of the channel with the least amount of radio traffic

> The smallest bar indicates the channel with the least amount of radio traffic.

- ▶ Select the **Start handwheel** button
- > The control restores the connection to the wireless handwheel.
- ▶ Select the **Properties** tab
- ▶ Select the **Select channel** button
- > The controls shows all available channel numbers.
- ▶ Select the number of the channel with the least amount of radio traffic
- ▶ Select the **END** button
- > The control saves the configuration.

34.3.4 Reactivating the handwheel

To reactivate the handwheel:

- ▶ Open the **Configuration of wireless handwheel** window
- ▶ Use the **Start handwheel** button to reactivate the wireless handwheel
- ▶ Select the **END** button

35

Touch Probes

35.1 Setting up touch probes

Application

The **Device configuration** window allows you to create and manage all the workpiece and tool touch probes of the control.

Touch probes with radio transmission can be created and managed only in the **Device configuration** window.

Related topics

- Creating a workpiece touch probe with cable or infrared transmission by using the touch probe table

  Further information: "Touch probe table tchprobe.tp", Page 1839
- Creating a tool touch probe with cable or infrared transmission by using the machine parameter **CfgTT** (no. 122700)

  Further information: "Machine parameters", Page 1958

Description of function

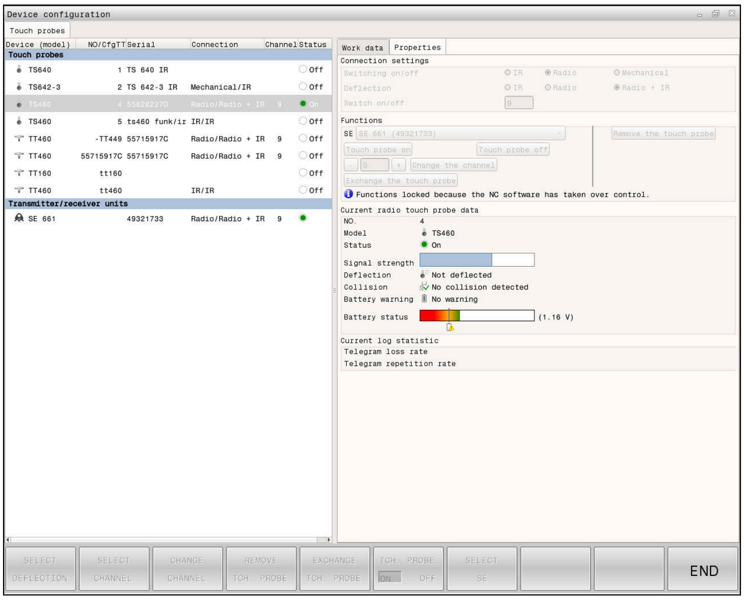
You open the **Device configuration** window in the **Machine settings** group of the **Settings** application. Double-tap or double-click the **Set up touch probes** menu item.

Further information: "Settings Application", Page 1911

Touch probes with radio transmission can be created and managed only in the **Device configuration** window.

In order for the control to recognize the touch probe with radio transmission, you will require an **SE 661** transceiver with EnDat interface.

You define the new values in the **Work data** area.



Areas of the Device configuration window

Touch probes area

In the **Touch probes** area, the control displays all of the defined workpiece and tool touch probes, as well as the transceiver units. All other areas provide detailed information about the selected entry.

Work data area

For a workpiece touch probe, the control displays the values from the touch probe table in the **Work data** area.

For a tool touch probe, the control displays the values from the machine parameter **CfgTT** (no. 122700).

You can select and edit the displayed values. Under **Touch probes**, the control displays information about the active value (e.g., selection options). You can change the values of the tool touch probes only after entering the code number 123.

Properties area

In the **Properties** area, the control displays the connection data and diagnostic functions.

For touch probes with radio connection, the control displays the following information in **Current radio touch probe data**:

Display	Meaning
NO.	Number in the touch probe table
Model	Type of touch probe
Status	Touch probe active or inactive
Signal strength	Display of the signal strength in the bar graphic The control shows the currently best-known connection as a complete bar
Deflection	Stylus deflected or not deflected
Collision	Collision or no collision recognized
Battery status	Display of the battery quality If the charge is less than the displayed bar, then the control outputs a warning.

The **Switching on/off** connection setting is preset based on the type of touch probe. Under **Deflection**, you can select how the touch probe is to transmit the signal when probing.

Deflection	Meaning
IR	Infrared probe signal
Radio	Radio probe signal
Radio + IR	The control selects the probe signal

> If you activate the touch probe's radio connection by using the connection setting **Switch on/off**, then the signal will be retained even after a tool change. You need to use this connection setting to deactivate the radio connection.

Buttons

The control provides the following buttons:

Button	Function
CREATE TS ENTRY	Create a new workpiece touch probe You define the new values in the **Work data** area.
CREATE TT ENTRY	Create a new tool touch probe You define the new values in the **Work data** area.
SELECT DEFLECTION	Select the probe signal
SELECT CHANNEL	Select the radio channel Select the channel with the best radio transmission and pay attention to overlaps with other machines or wireless handwheels.
CHANGE CHANNEL	Change the radio channel
REMOVE TCH. PROBE	Delete the touch probe data The control deletes the entry from the **Device configuration** window and from the touch probe table or the machine parameters.
EXCHANGE TCH. PROBE	Save a new touch probe in the current row The control automatically overwrites the serial number of the replaced touch probe with the new number.
SELECT SE	Select the SE transceiver
SELECT POWER	Select the strength of the infrared signal You only need to change the signal strength if there is interference.
SELECT POWER	Select the strength of the radio signal You only need to change the signal strength if there is interference.

Note

In the machine parameter **CfgHardware** (no. 100102), the machine manufacturer defines whether the control will show or hide the touch probes in the **Device configuration** window. Refer to your machine manual.

36

Embedded Workspace and Extended Workspace

36.1 Embedded Workspace (option 133)

Application

You use Embedded Workspace to operate a Windows PC and display its screen contents on the control's user interface. You use Remote Desktop Manager (option 133) to connect the Windows PC.

Related topics

- Remote Desktop Manager (option 133)

  Further information: "Remote Desktop Manager window (option 133)", Page 1944
- Using Extended Workspace to operate a Windows PC through an additional connected monitor

  Further information: "Extended Workspace", Page 1902

Requirements

- Established RemoteFX connection to the Windows PC through Remote Desktop Manager (option 133)
- Connection defined in the machine parameter **CfgRemoteDesktop** (no. 133500)

  In the optional machine parameter **connections** (no. 133501), the machine manufacturer enters the name of the RemoteFX connection.

  Refer to your machine manual.

Description of function

Embedded Workspace is available on the control as an operating mode and as a workspace. If the machine manufacturer does not define a name, then the operating mode and workspace are both named **RDP**.

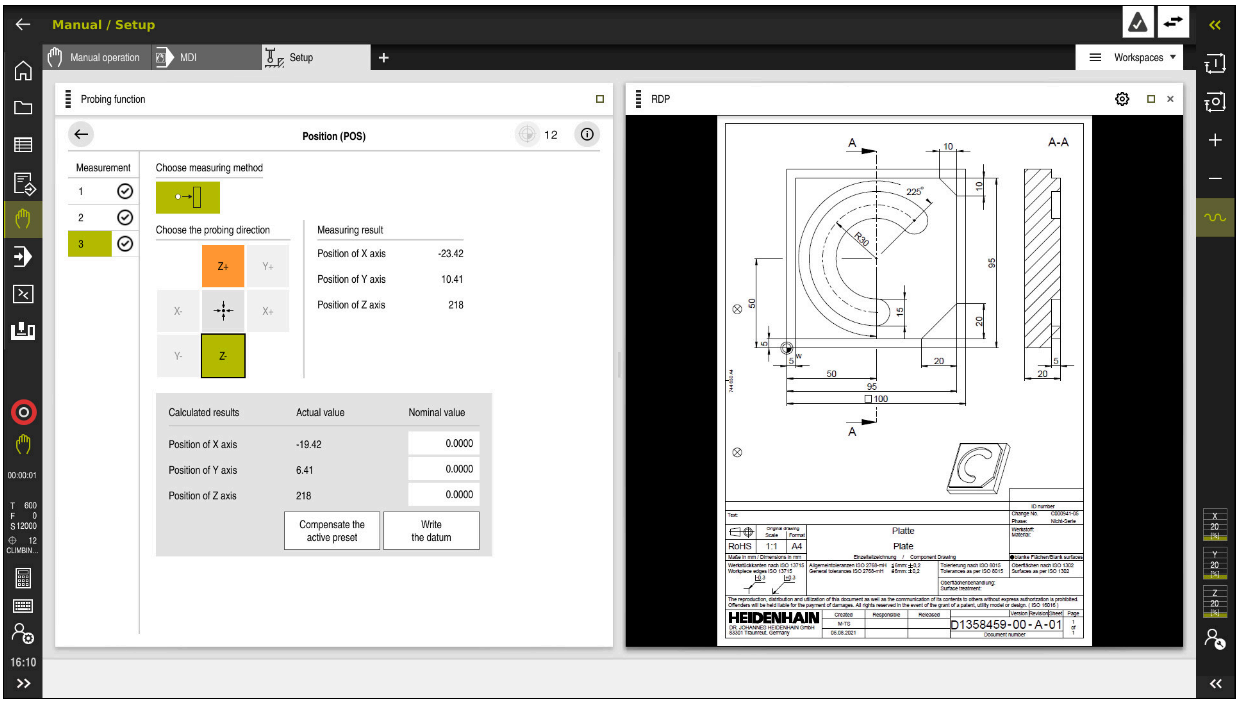
Entries cannot be made through the Windows PC as long as the RemoteFX connection is active. This avoids the problem of conflicting operation.

Further information: "Windows Terminal Service (RemoteFX)", Page 1945

If you open Embedded Workspace as an operating mode, the control displays a full-screen version of the Windows PC user interface in it.

If you open Embedded Workspace as a workspace, you can change the size and position of the workspace as you wish. The control rescales the user interface of the Windows PC after each modification.

Further information: "Workspaces", Page 92



Embedded Workspace as workspace with opened PDF file

RDP settings window

If Embedded Workspace is open as a workspace, you can open the **RDP settings** window.

The **RDP settings** window contains the following buttons:

Button	Meaning
Reconnect	If the control could not establish a connection to the Windows PC, for example due to a timeout, press this button to try again. The control can also display this button in the operating mode and the workspace.
Adjust resolution	With this button the control rescales the user interface of the Windows PC to the size of the workspace.

36.2 Extended Workspace

Application

With Extended Workspace you can use an additional attached monitor as a second screen of the control. That way you can use the additional monitor independently of the control's user interface and also to show the control's applications.

Related topics

- Using Embedded Workspace to operate a Windows PC within the control's user interface (option 133

  Further information: "Embedded Workspace (option 133)", Page 1900
- ITC hardware expansion

  Further information: "Hardware enhancements", Page 87

Requirement

- Additional attached monitor configured by the machine manufacturer as Extended Workspace

  Refer to your machine manual.

Description of function

Here are some functions you can perform with Extended Workspace:

- Opening files from the control (e.g., drawings)
- Opening windows from HEROS functions in addition to the control's user interface

  Further information: "HEROS menu", Page 1964
- Displaying and operating computers connected through Remote Desktop Managers (option 133)

  Further information: "Remote Desktop Manager window (option 133)", Page 1944

37

Integrated Functional Safety (FS)

Application

The safety concept of integrated functional safety (FS) for machines with HEIDENHAIN controls offers supplementary software safety functions in addition to the mechanical safety features of the machine. For example, the integrated safety concept automatically reduces the feed rate when you perform operations with open guard doors. The machine manufacturer can modify or expand the FS safety concept.

Requirements

- Software option 160 (Integrated functional safety (FS), basic version) or software option 161 (Integrated functional safety (FS), full version)
- Software options 162 to 166 or software option 169, if necessary

  Whether you need these software options depends on the machine's number of drives.
- The machine manufacturer must adapt the FS safety concept to the machine.

Description of function

Every machine tool user is exposed to certain risks. While protective devices can prevent access to dangerous locations, the user must also be able to work on the machine without this protection (e.g. guard door opened).

Safety functions

To ensure that the requirements for operator protection are met, integrated functional safety (FS) provides standardized safety functions. The machine manufacturer uses the standardized safety functions for implementing functional safety (FS) for the machine in question.

You can track the active safety functions in the axis status of functional safety (FS).

Further information: "Axis status menu item", Page 1907

Description	Meaning	Short description
SS0, SS1, SS1D, SS1F, SS2	Safe Stop	Safe stopping of drives using different methods
STO	Safe Torque Off	The power supply to the motor is interrupted. Provides protection against unexpected start of the drives
SOS	Safe Operating Stop	Safe operating stop. Provides protection against unexpected start of the drives
SLS	Safely Limited Speed	Safely limited speed. Prevents the drives from exceeding the specified speed limits when the protective door is opened
SLP	Safely Limited Position	Safely limited position. Monitors safe axes to keep them within the limit values of a defined area
SBC	Safe Brake Control	Dual-channel control of the motor holding brakes

Safety-related operating modes of functional safety (FS)

Functional safety (FS) of a control offers various safety-related operating modes. The safety-related operating mode with the lowest number has the highest safety level.

Depending on how the machine manufacturer implements them, the following safety-related operating modes are available:

Refer to your machine manual.
The machine manufacturer must adapt the safety-related operating modes to each machine.

Icon	Safety-related operating mode	Short description
SOM 1	Operating mode **SOM_1**	Safe operating mode 1: Automatic mode, production mode
SOM 2	Operating mode **SOM_2**	Safe operating mode 2: Set-up mode
SOM 3	Operating mode **SOM_3**	Safe operating mode 3: Manual intervention; only for qualified operators
SOM 4	Operating mode **SOM_4** This function must be enabled and adapted by the machine manufacturer.	Safe operating mode 4: Advanced manual intervention, process monitoring, only for qualified users

Functional safety (FS) in the Positions workspace

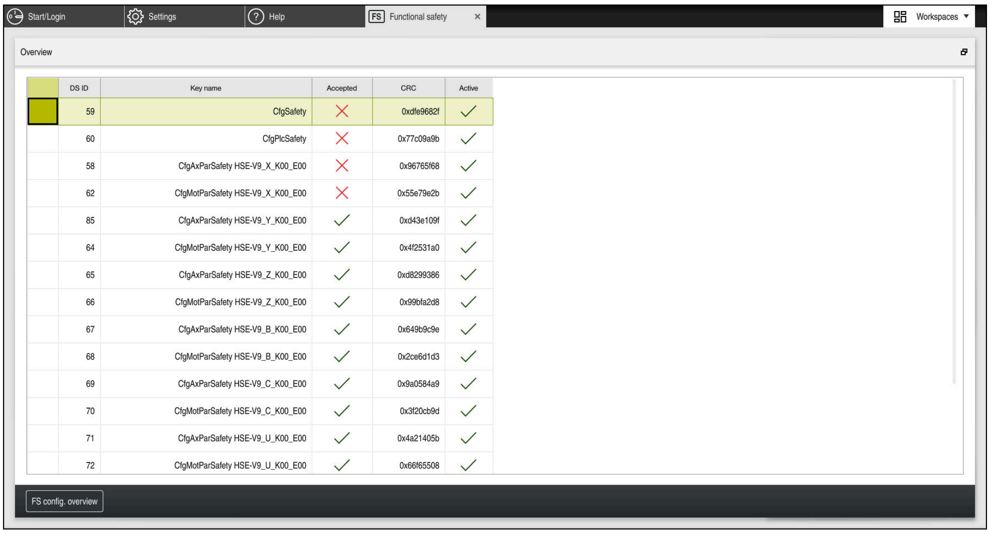
On a control with functional safety (FS), the control displays the monitored operating statuses of the speed **S** and feed rate **F** in the **Positions** workspace. If a safety function is triggered while in a monitored state, the control stops the feed movement and the spindle or reduces the speed (e.g., if a guard door is opened).

Further information: "Axis display and position display", Page 142

Functional safety application

⚙	Refer to your machine manual. The machine manufacturer configures the safety functions in this application.

In the **Functional safety** application in the **Home** operating mode, the control provides information about the status of the individual safety functions. In this application you can see whether individual safety functions are active and have been accepted by the control.

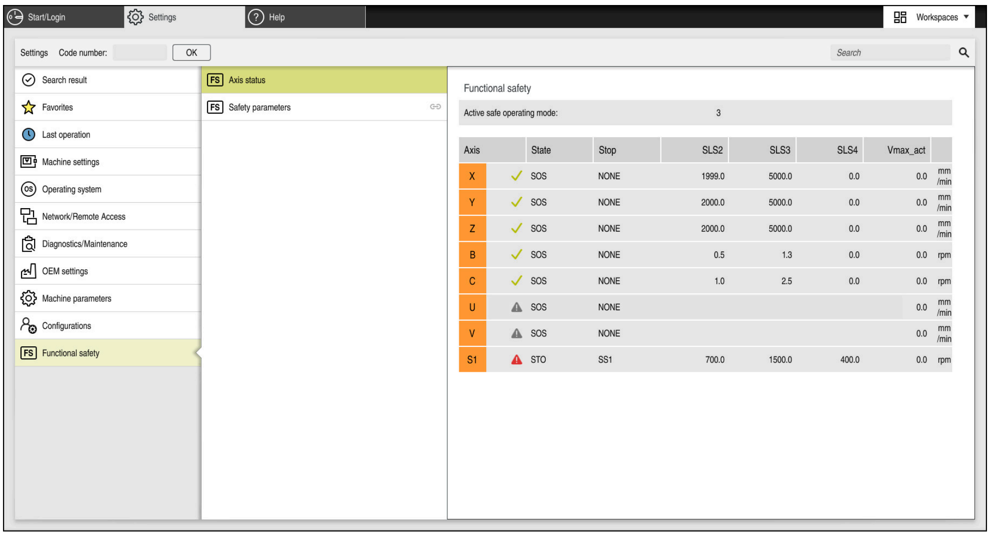


Functional safety application

Axis status menu item

In the **Axis status** menu item of the **Settings** application, the control provides the following information about the status of the individual axes:

Field	Meaning
Axis	Configured axes of the machine
State	Active safety function
Stop	Stop reaction **Further information:** "Functional safety (FS) in the Positions workspace", Page 1905
SLS2	Maximum speed or feed-rate values for **SLS** in the **SOM_2** operating mode
SLS3	Maximum speed or feed-rate values for **SLS** in the **SOM_3** operating mode
SLS4	Maximum speed or feed-rate values for **SLS** in the **SOM_4** operating mode This function must be enabled and adapted by the machine manufacturer.
Vmax_act	Currently valid speed or feed rate limit These are either values from the **SLS** settings or from the SPLC If values are greater than 999 999, the control displays **MAX**.



Axis status menu item in the **Settings** application

Test status of the axes

In order for the control to ensure safe operation of the axes, it checks all monitored axes when the machine is switched on.

The control checks whether the position of an axis matches the position directly after shutdown. If a deviation is detected, the control marks the respective axis in the position display with a red warning triangle.

If checking of individual axes fails when starting the machine, you can check the axes manually.

Further information: "Checking axis positions manually", Page 1909

The control indicates the test status of the individual axes with the following icons:

Icon	Meaning
	The axis has been tested or does not need to be tested.
	The axis has not been tested, but must be tested to ensure safe operation. **Further information:** "Checking axis positions manually", Page 1909
	The axis is not monitored by functional safety (FS) or is not configured as a safe axis.

Feed-rate limiting with functional safety (FS)

> Refer to your machine manual.
> This function must be adapted by your machine manufacturer.

With the **FMAX** button you can prevent the SS1 reaction for safe stopping of drives when the guard door is opened.

By pressing the **FMAX** button, the control limits the speed of the axes and of the spindle(s) to the values defined by the machine manufacturer. The limitation depends on the active safety-related SOM_x operating mode. You can select the safety-related operating mode with the keylock switch.

> In the safety-related operating mode SOM_1, the control stops the axes and spindles when the guard door is opened.

37.1 Checking axis positions manually

Refer to your machine manual.
This function must be adapted by your machine manufacturer.
The machine manufacturer defines the test position.

To check the position of an axis:

- Select the **Manual** operating mode
- Select **Approach test position**
- > The control displays the axes that have not been tested in the **Positions** workspace.
- Select the desired axis in the **Positions** workspace
- Press the **NC start** key
- > The axis moves to the test position.
- > After the test position has been reached, the control issues a message.
- Press the **permissive button** on the machine operating panel
- > The control displays the axis as a tested axis.

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision while approaching the test positions.

- If necessary, move to a safe position before approaching the test positions
- Watch out for possible collisions

Notes

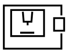
- Machine tools with HEIDENHAIN controls may be equipped with integrated functional safety (FS) or with external safety. This chapter refers exclusively to machines with integrated functional safety (FS).
- The machine manufacturer defines the behavior of speed-controlled FS-NC axes while the guard door is open in the machine parameter **speedPosCompType** (no. 403129). The machine manufacturer can e.g. allow switching-on of the spindle and thus enable scratching of the workpiece while the guard door is open. Refer to your machine manual.

38

Settings Application

38.1 Overview

The **Settings** application includes the following groups with menu items:

Icon	Category	Menu item
	**Machine settings**	■ **Machine settings** **Further information:** "Machine settings menu item", Page 1914 ■ **General information** **Further information:** "General information menu item", Page 1916 ■ **SIK** **Further information:** "SIK menu item", Page 1917 ■ **Machine times** **Further information:** "Machine times menu item", Page 1918 ■ **Set up touch probes** **Further information:** "Setting up touch probes", Page 1896 ■ **Set up wireless handwheel** **Further information:** "HR 550FS wireless handwheel", Page 1889
	**Operating system**	■ **PKI Admin** Manage certificates for the control (e.g., for **OPC UA NC Server**) **Further information:** "OPC UA NC Server (options 56 to 61)", Page 1931 ■ **Date/Time** **Further information:** "Adjust system time window", Page 1919 ■ **Language/Keyboards** **Further information:** "Conversational language of the control", Page 1919 ■ **About HEROS** **Further information:** "Information on licensing and use", Page 82 ■ **SELinux** **Further information:** "SELinux security software", Page 1921 ■ **UserAdmin** Currently no function ■ **Current User** Currently no function ■ **Touchscreen configuration** You can choose the touchscreen's sensitivity and whether touchpoints should be shown or hidden.

Icon	Category	Menu item
	**Network/Remote Access**	■ **Shares** **Further information:** "Network drives on the control", Page 1922 ■ **Network** **Further information:** "Ethernet interface", Page 1925 ■ **Printer** **Further information:** "Printers", Page 1937 ■ **DNC** **Further information:** "DNC menu item", Page 1936 ■ **OPC UA** **Further information:** "OPC UA NC Server (options 56 to 61)", Page 1931 ■ **VNC** **Further information:** "VNC menu item", Page 1940 ■ **Remote Desktop Manager** **Further information:** "Remote Desktop Manager window (option 133)", Page 1944 ■ **Real VNC Viewer** Define settings for external software accessing the control (e.g., for maintenance purposes); for network specialists ■ **Firewall** **Further information:** "Firewall", Page 1950
	**Diagnostics/Maintenance**	■ **Terminal program** Enter and execute console commands ■ **HeLogging** Define settings for internal diagnostic files ■ **Portscan** **Further information:** "Portscan", Page 1953 ■ **perf2** Check processor load and process load ■ **RemoteService** **Further information:** "Remote servicing", Page 1954 ■ **NC/PLC Restore** **Further information:** "Backup and restore", Page 1955 ■ **TNCdiag** **Further information:** "TNCdiag", Page 1957 ■ **NC/PLC Backup** **Further information:** "Backup and restore", Page 1955 ■ **Touchscreen cleaning** The control disables the touchscreen for 90 seconds.
	**OEM settings**	Settings for the machine manufacturer
	**Machine parameters**	The group contains machine parameters that can be edited, depending on your rights (e.g., **MPs for setters**). **Further information:** "Machine parameters", Page 1958

Icon	Category	Menu item
FS	Functional safety	■ **Axis status** **Further information:** "Axis status menu item", Page 1907 ■ **Safety parameters** **Further information:** "Functional safety application", Page 1906

38.2 Code numbers

Application

The top part of the **Settings** application contains the **Code number:** input field and the **unit of measure** switch. These settings are possible and accessible from every group.

Description of function

You can enable the following functions or areas with code numbers:

Code number	Function
123	Editing machine-specific user parameters **Further information:** "Machine parameters", Page 1958
555343	Special functions for programming with variables **Further information:** "Variable Programming", Page 1267

> The control indicates whether the caps lock key is pressed during entry. This helps to avoid incorrect entries.

38.3 Machine settings menu item

Application

In the **Machine settings** menu item of the **Settings** application, you can define the settings for simulation and program run.

Related topics

- Graphic settings for simulation
  Further information: "Simulation settings window", Page 1411

Description of function

Unit of measure area

In the **Unit of measure** area you can choose between mm and inch.

- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), the value is displayed to 4 decimal places

If the display in inches is active, the control also displays the feed rate in inches/min. In an inch-based program, you must multiply the feed rate by 10.

Channel settings

The control displays the channel settings separately for the **Editor** operating mode and the **Manual** and **Program Run** operating modes.

You can define the following settings:

Setting	Meaning
Active kinematics	Use the **Active kinematics** function to change the kinematics model for the machine and the simulation. This way you can test NC programs that, for example, are programmed for other machines. The control offers a selection menu with all available kinematics models. The machine manufacturer defines which kinematics models you can choose. The control displays the active kinematics model in the **Machine** mode of the **Simulation** workspace.
Generate tool-usage file	The control uses the tool-usage file to check tool usage. **Further information:** "Tool usage test", Page 284 You select when the control should generate a tool-usage file: ■ **Never** The control does not generate a tool-usage file. ■ **Once** The next time you simulate or run an NC program, the control will generate a tool-usage file once. ■ **Always** When you simulate or run an NC program, the control will generate a tool-usage file each time.

Traverse limits

Use the **Traverse limits** function to limit the possible traverse path of an axis. You can define traverse limits for each axis (e.g., to protect an indexing head from collision).

The **Traverse limits** function consists of a table with the following contents:

Column	Meaning
Axis	The TNC displays each axis of the active kinematics model in a row.
Status	If you have defined one or both limits, the control displays the contents **Valid** or **Invalid**.
Lower limit	You define the lower traverse limit of the axis in this column. You can enter up to four decimal places.
Upper limit	You define the upper traverse limit of the axis in this column. You can enter up to four decimal places.

The defined traverse limits are valid across power cycles of the control, until you delete all values from the table.

The following general conditions apply to the traverse limit values:

- The lower limit must be smaller than the upper limit.
- The upper and lower limit may not both equal 0.

Other conditions apply to traverse limits for modulo axes.

Further information: "Notes on software limit switches for modulo axes", Page 1218

Notes

NOTICE
Danger of collision! You can also select any stored kinematics model as the active machine kinematics. The control then executes all manual movements and machining operations using the selected kinematics. All subsequent axis movements pose a risk of collision! ▶ Use the **Active kinematics** function for the simulation only ▶ Use the **Active kinematics** function for selecting the active machine kinematics only if required

- In the optional machine parameter **enableSelection** (no. 205601), the machine manufacturer defines for each kinematics model whether the **Active kinematics** function can be selected.
- You can open the tool-usage file in the **Tables** operating mode.
  Further information: "Tool usage file", Page 1845
- If the control has generated a tool-usage file for an NC program, then there are contents in the **T usage order** and **Tooling list** tables (option 93).
  Further information: "T usage order (option 93)", Page 1847
  Further information: "Tooling list (option 93)", Page 1849

38.4 General information menu item

Application

In the **General information** menu item of the **Settings** application, the control provides the information about the control and the machine.

Description of function

Version information area

The control displays the following information:

Sub-area	Meaning
HEIDENHAIN	■ **Control model** Designation of the control (managed by HEIDENHAIN) ■ **NC-SW** Number of the NC software (managed by HEIDENHAIN) ■ **NCK** Number of the NC software (managed by HEIDENHAIN)
PLC	**PLC-SW** Number or name of the PLC software (managed by the machine manufacturer)

The machine manufacturer can add further software numbers (e.g., that of a connected camera).

Info about machine manufacturer area

The control shows the contents of the optional machine parameter **CfgOemInfo** (no. 131700). The control displays this area only if the machine manufacturer defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 1932

Machine information area (options 56 to 61)

The control shows the contents of the optional machine parameter **CfgMachineInfo** (no. 131600, options 56 to 61). The control displays this area only if the machine operator defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 1932

38.5 SIK menu item

Application

Use the **SIK** menu item of the **Settings** application to view control-specific information (e.g., the serial number and the available software options).

Related topics

- Software options on the control

  Further information: "Software options", Page 74

Description of function

SIK information area

The control displays the following information:

- **Serial number**
- **Control model**
- **Performance class**
- **Features**
- **Status**

OEM key area

In the **OEM key** area the machine manufacturer can define a manufacturer-specific password for the control.

General key area

In the **General key** area the machine manufacturer can enable all software options once for a period of 90 days (e.g., for testing).

The control indicates the status of the general key:

Status	Meaning
NONE	The general key has not yet been used for this software version.
dd.mm.yyyy	Date up to which all software options will be available. Once the general key has expired, it cannot be used again.
EXPIRED	The general key has expired for this software version.

If the software version of the control is increased (e.g., by an update), then the **General key** can be used again.

Software options area

In the **Software options** area the control shows all available software options in a table.

Column	Meaning
#	Number of the software option
Option	Name of the software option
Expiration date	The machine manufacturer can enable software options for a limited time. In this case the control shows in this column the date through which the software option is available.
	The machine manufacturer uses the **Set** button to enable a software option. For enabled software options the control displays the text **Enabled**.

38.5.1 Viewing of software options

To view enabled software options on the control:

- Select the **Home** operating mode
- Select the **Settings** application
- Select **Machine settings**
- Select **SIK**
- Navigate to the **Software options** area
- > For enabled software options the control displays the text **Enabled** at the end of the row.

Definition

Abbreviation	Definition
SIK (System Identification Key)	**SIK** is the designation of the plug-in board for the control hardware. Each control can clearly be identified by the serial number of the **SIK**.

38.6 Machine times menu item

Application

In the **Machine times** menu item of the **Settings** application, the control shows the run times since initial setup.

Related topics

- Date and time of the control
  Further information: "Adjust system time window", Page 1919

Description of function

The control displays the following machine times:

Machine time	Meaning
Control on	Run time of the control since being put into service
Machine on	Run time of the machine tool since being put into service
Program Run	Run time of all program runs since being put into service

Refer to your machine manual.
The machine manufacturer can define up to 20 additional run times.

38.7 Adjust system time window

Application

In the **Adjust system time** window you can set the time zone, date and time manually or by means of an NTP server synchronization.

Related topics

- Run times of the machine tool
  Further information: "Machine times menu item", Page 1918

Description of function

The **Date/Time** menu item opens the **Adjust system time** window. The menu item is in the **Operating system** group of the **Settings** application.

The **Adjust system time** window consists of the following areas:

Area	Function
Set the time manually	Activate this check box to define the following data: ■ Year ■ Month ■ Day ■ Time
Synchronize the time over NTP server	If you activate this check box, the control will automatically synchronize the system time with the defined NTP server. You can add a server with a host name or a URL.
Time zone	You can select your time zone from a list.

38.8 Conversational language of the control

Application

You use the **helocale** window to change the conversational language of the HEROS operating system and the machine parameters to change the NC conversational language of the control's user interface.

The HEROS conversational language only changes after a restart of the control.

Related topics

- Machine parameters of the control

  Further information: "Machine parameters", Page 1958

Description of function

You can't define two different conversational languages for the operating system and control.

The **Language/Keyboards** menu item opens the **helocale** window. The menu item is in the **Operating system** group of the **Settings** application.

The **helocale** window consists of the following areas:

Area	Function
Language	Choose the HEROS conversational language from a selection menu Only if the machine parameter **applyCfgLanguage** (no. 101305) is defined as **FALSE**.
Keyboards	Select the language layout of the keyboard for HEROS functions

38.8.1 Changing the language

By default, the control assumes the NC conversational language for the HEROS conversational language.

To change the NC conversational language:

- ▶ Select the **Settings** application
- ▶ Enter the code number 123
- ▶ Select **OK**
- ▶ Select **Machine parameters**
- ▶ Double-tap or double-click **MPs for setters**
- > The control opens the **MPs for setters** application.
- ▶ Navigate to the machine parameter **ncLanguage** (no. 101301)
- ▶ Select the desired language

- ▶ Select **Save**
- > The control opens the **Configuration data changed. All changes.** window.

- ▶ Select **Save**
- > The control opens the notification menu and displays a "Question type" error.

- ▶ Select **CLOSE CONTROL**
- > The control restarts.
- > Once the control has restarted, the NC conversational language and the HEROS conversational language are changed.

Note

Use the machine parameter **applyCfgLanguage** (no. 101305) to define whether the control assumes the setting for the NC conversational language for the HEROS conversational language.

- **TRUE** (default): The control assumes the NC conversational language. You can change the language only in the machine parameters.

  Further information: "Changing the language", Page 1920
- **FALSE**: The control assumes the HEROS conversational language. You can change the language only in the **helocale** window.

38.9 SELinux security software

Application

SELinux is an extension for Linux-based operating systems in the sense of Mandatory Access Control (MAC). The security software protects the system against the execution of unauthorized processes or functions, i.e. viruses and other malicious software.

The machine manufacturer defines the **SELinux** settings in the **Security Policy Configuration** window.

Related topics

- Security settings with firewall

  Further information: "Firewall", Page 1950

Description of function

The **SELinux** menu item opens the **Security Policy Configuration** window. The menu item is in the **Operating system** group of the **Settings** application.

The access control of **SELinux** is regulated as follows by default:

- The control executes only programs that are installed with the HEIDENHAIN NC software.
- Only explicitly selected programs can modify safety-relevant files, such as **SELinux** system files or HEROS boot files.
- New files created by other programs may not be run.
- USB data carriers can be deselected.
- Only two processes can run new files:
  - Software update: A software update from HEIDENHAIN can replace or modify system files.
  - SELinux configuration: The configuration of **SELinux** with the **Security Policy Configuration** window is usually password-protected by the machine manufacturer (refer to the relevant machine manual).

Note

HEIDENHAIN recommends using **SELinux** as additional protection against attacks from outside the network.

Definition

Abbreviation	Definition
MAC (mandatory access control)	MAC means that the control performs only explicitly permitted actions. **SELinux** is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can be performed only if the standard functions and access control of **SELinux** permit it.

38.10 Network drives on the control

Application

Use the **Mount Setup** window to connect network drives to the control. If a network drive is connected to the control, the control displays additional drives in the navigation column of the file management.

Related topics

- File management

  Further information: "File management", Page 1066
- Network settings

  Further information: "Ethernet interface", Page 1925

Requirements

- Existing network connection
- Control and computer in same network
- Path and access data of drive to be connected are known

Description of function

The **Shares** menu item opens the **Mount Setup** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

You can also open the window with the **Mount network share** button of the **Files** operating mode.

Further information: "File management", Page 1066

You can define any number of network drives, but only seven can be connected at a time.

Network drive area

In the **Network drive** area, the control shows a list of all defined network drives, as well as the status of each drive.

The control displays the following buttons:

Button	Meaning
Mount	Connect a network drive The control selects the check box in the **Mount** column if an active connection exists.
Unmount	Disconnect a network drive
Auto	Automatically connect the network drive when the control is booting. The control selects the check box in the **Auto** column if an active automatic connection exists.
Add	Define a new connection **Further information:** "Mount assistant window", Page 1924
Remove	Delete an existing connection
Copy	Copy connection **Further information:** "Mount assistant window", Page 1924
Edit	Edit the connection settings **Further information:** "Mount assistant window", Page 1924
Private network drive	User-specific connection if user administration is active The control selects the check box in the **Privat** column if a user-specific connection exists.

Status Log area

In the **Status Log** area, the control shows status information and error messages about connections.

Use the **Clear** button to delete the contents of the **Status Log** area.

Mount assistant window

In the **Mount assistant** window you define the settings for a connection with a network drive.

The **Add**, **Copy** and **Edit** buttons open the **Mount assistant** window.

The **Mount assistant** window contains tabs with the following settings:

Tab	Setting
Drive name	■ **Drive name:** Network drive name in the file management of the control The control permits only capitals, with a colon (**:**) at the end. ■ **Private network drive** When user administration is active, the connection is only visible to the user who created it.
Share type	Transfer protocol ■ **Windows share (CIFS/SMB) or Samba server** ■ **UNIX share (NFS)**
Server and Share	■ **Server name:** Server name or IP address ■ **Share name:** Directory accessed by the control
Automount	**Connect automatically (not possible with the "Ask for password?" option)** The control connects the network drive automatically during the starting process.
User name and password (only with Windows share)	■ **Single Sign On** When user administration is active, the control automatically connects an encrypted network drive when the user logs in. ■ **Windows user name:** ■ **Ask for password? (not possible with the "Connect automatically" option)** Select whether a password is required upon connecting. ■ **Password** ■ **Password verification**
Mounting options	**Parameters for mount option "-o":** Auxiliary parameters for the connection **Further information:** "Examples of Mounting options", Page 1925
Check	The control displays a summary of the defined settings. You can check the settings and save them with **Apply**.

Examples of Mounting options

Enter options without a space, only separated by a comma

Options for SMB

Example	Meaning
domain=xxx	Name of the domain HEIDENHAIN recommends not to write the domain into the user name, but as an option.
vers=2.1	Protocol version

Options for NFS

Example	Meaning
rsize=8192	Packet size in bytes for data reception Input: **512...8192**
wsize=4096	Packet size in bytes for data transmission Input: **512...8192**
soft,timeo=3	Conditional Mount Time in tenths of a second after which the control will try to connect again
sec=ntlm	Authentication method ntlm Use this option if the control displays the **Permission denied** error message upon connecting.
nfsvers=2	Protocol version

Notes

- Have a network specialist configure the control.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

38.11 Ethernet interface

Application

The control is provided with an Ethernet interface as a standard feature so that you can integrate it into a network.

Related topics

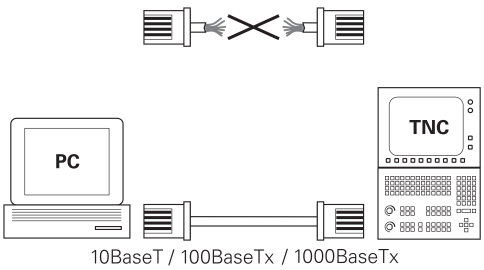
- Firewall settings
  Further information: "Firewall", Page 1950
- Network drives on the control
  Further information: "Network drives on the control", Page 1922
- External access
  Further information: "DNC menu item", Page 1936

Description of function

The control transfers data via the Ethernet interface using the following protocols:

- **CIFS** (common internet file system) or **SMB** (server message block)
  The control supports versions 2, 2.1 and 3 of these protocols.
- **NFS** (network file system)
  The control supports versions 2 and 3 of this protocol.

Connection options



You can integrate the Ethernet interface of the control into the network or connect it directly to a PC through the RJ45 connection X26. The connection is electrically isolated from the control electronics.

Use a Twisted Pair cable to connect the control to your network.

> The maximum cable length permissible between the control and a node depends on the quality grade of the cable, the sheathing, and the type of network.

Ethernet connection icon

Icon	Meaning
	Ethernet connection The control displays the icon at the bottom right in the taskbar. **Further information:** "Taskbar", Page 1967 When you click the icon, the control opens a pop-up window. The pop-up window contains the following information and functions: ▪ Connected networks You can disconnect the network connection. Select the network name to reconnect. ▪ Available networks ▪ VPN connections Currently no function

Notes

- Protect your data and the control by running the machines in a secure network.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

38.11.1 Network settings window

Application

In the **Network settings** window you define the settings for the control's Ethernet interface.

> Have a network specialist configure the control.

Related topics

- Network configuration

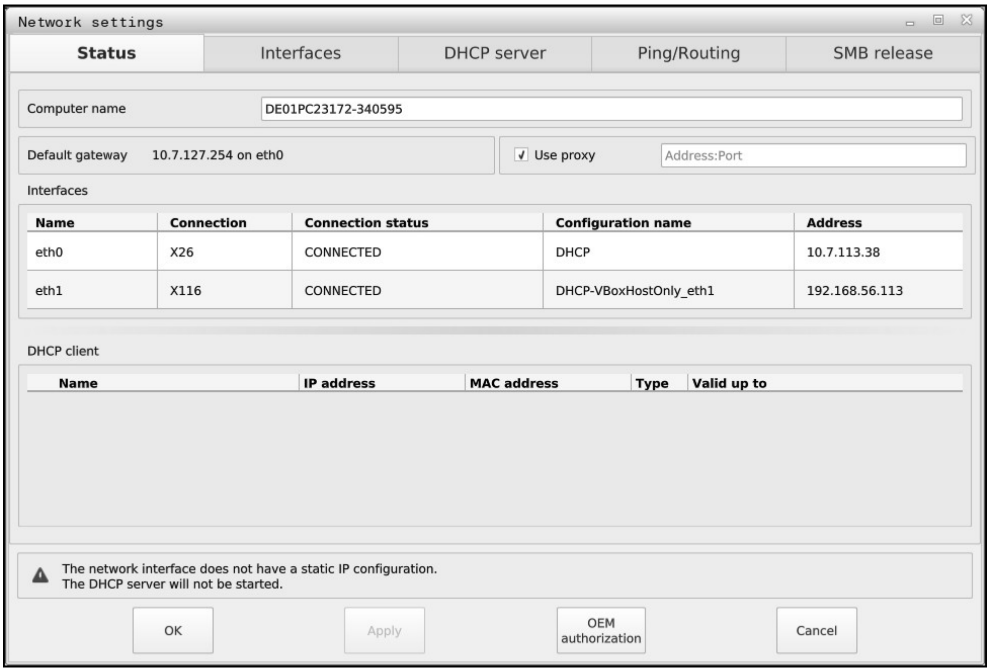
  Further information: "Network configuration with Advanced Network Configuration", Page 1974
- Firewall settings

  Further information: "Firewall", Page 1950
- Network drives on the control

  Further information: "Network drives on the control", Page 1922

Description of function

Open the **Network settings** window via the **Network** menu item. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.



Network settings window

Status tab

The **Status** tab contains the following information and settings:

<table>
<tr><th>Domain</th><th>Information or Setting</th></tr>
<tr><td>Computer name</td><td>The control displays the name under which the control is visible in the company network. You can change the name.</td></tr>
<tr><td>Default gateway</td><td>The control shows the default gateway and the Ethernet interface being used.</td></tr>
<tr><td>Use proxy</td><td>You can define the address and the port of a proxy server in the network.</td></tr>
<tr><td>Interfaces</td><td>The control shows an overview of available Ethernet interfaces. If there is no network connection, the table is empty.
The control displays the following information in the table:
Name, e.g. eth0
Connection, e.g. X26
Connection status, e.g. CONNECTED
Configuration name, e.g. DHCP
Address, e.g. 10.7.113.10
Further information: "Interfaces tab", Page 1929</td></tr>
<tr><td>DHCP client</td><td>The control displays an overview of the devices that have received a dynamic IP address in the machine network. If there are no connections to other network components of the machine network, the table is empty.
The control displays the following information in the table:
Name
Host name and connection status of the device.
The control shows the following connection status:
Green: Connected
Red: No connection
IP address
Dynamically assigned network address of the device
MAC address
Physical address of the device
Type
Type of connection
The control displays the following connection types:
TFTP
DHCP
Valid up to
Time until which the IP address is valid without being renewed
The machine manufacturer can make settings for these devices. Refer to your machine manual.</td></tr>
</table>

Interfaces tab

The control displays the available Ethernet interfaces on the **Interfaces** tab.

The **Interfaces** tab contains the following information and settings:

Column	Information or Setting
Name	The control displays the name of the Ethernet interface. You can activate or deactivate the connection by means of a switch.
Connection	The control displays the number of the network connection.
Connection status	The control displays the connection status of the Ethernet interface. The following connection statuses may be displayed: ■ **CONNECTED** Connected ■ **DISCONNECTED** Connection separated ■ **CONFIGURING** The IP address is being fetched from the server ■ **NOCARRIER** No cable present
Configuration name	You can execute the following functions: ■ Select a profile for the Ethernet interface In the factory default setting, two profiles are available: ■ **DHCP-LAN**: Settings for the standard interface for a standard company network ■ **MachineNet**: Settings for the second, optional Ethernet interface; for configuration of the machine network **Further information:** "Network configuration with Advanced Network Configuration", Page 1974 ■ Reconnect the Ethernet interface with **Reconnect** ■ Edit the selected profile **Further information:** "Network configuration with Advanced Network Configuration", Page 1974

The control additionally offers the following functions:

- **Set standard values**

  The control resets all settings. The control restores the profiles that are available in the factory default setting.
- **Configuration name**

  You can add, edit or remove profiles for the network connection.

  > If you have changed the profile of an active connection, the control will not update the profile being used. Reconnect the corresponding interface with **Reconnect**.

  The control exclusively supports the **Ethernet** connection type.

  Further information: "Network configuration with Advanced Network Configuration", Page 1974

DHCP server tab

The machine manufacturer can use the **DHCP server** tab in the control to configure a DHCP server in the machine network. Using this server, the control can establish connections with other network components of the machine network, e.g. with industrial computers.

Refer to your machine manual.

Ping/Routing tab

You can check the network connection on the **Ping/Routing** tab.

The **Ping/Routing** tab contains the following information and settings:

Domain	Information or Setting
Ping	**Address:Port** and **Address:** You can enter the IP address of the computer and possibly the port number for checking the network connection. Entry: Four numerical values separated by dots, possibly a port number separated by a colon e.g. **10.7.113.10:22** As an alternative, you can enter the name of the computer whose connection you want to check. Starting and stopping the test ■ **Start** button: starts the test The control displays status information in the ping field. ■ **Stop** button: stops the test
Routing	The control displays status information of the operating system about the current routing for network administrators.

SMB share tab

The **SMB share** tab is included only in connection with a VBox programming station.

When the check box is active, the control releases areas or partitions protected by a code number for the Explorer of the Windows PC used, e.g. **PLC**. You can activate or deactivate the check box only by using the machine manufacturer code number.

In the **TNC VBox Control Panel**, select a drive letter within the **NC share** tab for displaying the selected partition and then connect the drive with **Connect**. The host displays the partitions of the programming station.

Further information: Programming station for milling controls

You download the documentation together with the programming station software.

Notes

- Preferably restart the control after making changes in the network settings.
- The HEROS operating system manages the **Network settings** window. If you change the HEROS conversational language, you must restart the control for the change to take effect.

  Further information: "Conversational language of the control", Page 1919

38.12 OPC UA NC Server (options 56 to 61)

38.12.1 Fundamentals

Open Platform Communications Unified Architecture (OPC UA) describes a collection of specifications. These specifications are used to standardize machine-to-machine communication (M2M) in the field of industrial automation. OPC UA enables the data exchange across operating systems between products from different manufacturers, e.g. between a HEIDENHAIN control system and third-party software. Thus, OPC UA has become the data exchange standard for secure, reliable, manufacturer- and platform-independent industrial communication over the last years.

In 2016, the German Federal Office for Information Security (BSI) published a security analysis related to **OPC UA**. The specification analysis performed by the BSI determined that **OPC UA** provides a high level of security as compared to most other industrial protocols.

HEIDENHAIN follows the BSI recommendations and provides SignAndEncrypt, which exclusively features up-to-date IT security profiles. For this purpose, OPC UA-based industrial applications and the **OPC UA NC Server** exchange certificates for authentication. In addition, any transferred data is encrypted. This effectively prevents messages between the communication partners from being intercepted or altered.

Application

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The **OPC UA NC Server** allows you to access the data and functions of the HEIDENHAIN NC information model exposed in the server address space.

Related topics

- **Information Model** interface documentation with the specification of the **OPC UA NC Server** in English

  ID: 1309365-xx or **OPC UA NC Server interface documentation**

Requirements

- OPC UA NC Server (software options 56 to 61)

  For OPC UA-based communication, the HEIDENHAIN control provides the **OPC UA NC Server**. For each OPC UA client to be connected, you need one of the six available software options (56 to 61).
- Firewall configured

  Further information: "Firewall", Page 1950
- OPC UA client supports the **Security Policy** and the authentication method of the **OPC UA NC Server**:
  - **Security Mode: SignAndEncrypt**
  - **Algorithm: Basic256Sha256**
  - **User Authentication: X509 certificates**

Description of function

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The control supports the following OPC UA functions:

- Write and read variables
- Subscribe to value changes
- Run methods
- Subscribe to events
- File system access to the **TNC:** drive
- File system access to the **PLC:** drive (a corresponding permission is required)

Machine parameters in conjunction with OPC UA

The **OPC UA NC Server** enables OPC UA client applications to query general machine information, such as the year of construction of the machine or its location.

The following machine parameters are available for the digital identification of your machine:

- For users: **CfgMachineInfo** (no. 131700)

  Further information: "Machine information area (options 56 to 61)", Page 1917
- For the machine tool builder: **CfgOemInfo** (no. 131600)

  Further information: "Info about machine manufacturer area", Page 1917

Access to directories

The **OPC UA NC Server** enables read and write access to the **TNC:** and **PLC:** drives.

The following actions are permitted:

- Creation and deletion of folders
- Reading, editing, copying, moving, creating, and deleting of files.

While the NC software is running, the files referenced in the following machine parameters are locked against write access:

- Tables referenced by the machine manufacturer in the machine parameter **CfgTablePath** (no. 102500)
- Files referenced by the machine manufacturer in the machine parameter **dataFiles** (no. 106303, branch **CfgConfigData** no. 106300)

The **OPC UA NC Server** enables access to the control even if the NC software is switched off. You can e.g. transfer automatically created service files at any time as long as the operating system is active.

NOTICE

Caution: potential damage to property!

The control does not automatically back up the files before editing or deletion. Files that are missing cannot be restored. The removal or editing of system-relevant files, such as the tool table, can negatively affect the control functions.

- ▶ System-relevant files must be edited only by authorized specialists

Required certificates

The **OPC UA NC Server** requires three different types of certificates. The server and the client need two of them, the application instance certificates, in order to establish a secure connection. The third certificate (user certificate) is required for authorization and for starting a session with specific user permissions.

The control automatically generates a two-level certificate chain referred to as the **Chain of Trust** for the server. This certificate chain consists of a self-signed root certificate (including a **revocation list**) and a certificate for the server that is created on the basis of the root certificate.

The client certificate must be added on the **Trusted** tab of the **PKI Admin** function.

All other certificates should be added on the **Issuers** tab of the **PKI Admin** function for verification of the entire certificate chain.

User certificate

The control uses the HEROS functions **Current User** or **UserAdmin** for administration of the user certificate. When you initiate a session, the rights of the associated internal user are active.

To assign a user certificate to a user:

- Open the **Current User** HEROS function
- Select **SSH keys and certificates**
- Press the **Import certificate** soft key
- > The control opens a pop-up window.
- Select the certificate
- Select **Open**
- > The control imports the certificate.
- Press the **Use for OPC-UA** soft key

Self-generated certificates

You can also create and import all of the required certificates yourself.

Self-generated certificates must fulfill the following requirements:

- General requirements
  - File format: *.der
  - Signature with hash SHA256
  - Validity period of at most 5 years is recommended
- Client certificates
  - Host name of the client
  - Application URI of the client
- Server certificates
  - Host name of the control
  - Application URI of the server according to the following structure: urn:<hostname>/HEIDENHAIN/OpcUa/NC/Server
  - Validity period of 20 years maximum

Note

OPC UA is a manufacturer/platform-independent, open communication standard. For this reason, an OPC UA client SDK is not included in the **OPC UA NC Server**.

38.12.2 OPC UA menu item (options 56 to 61)

Application

In the **OPC UA** menu item of the **Settings** application, you can set up the connections to the control and check the status of the OPC UA connections.

Description of function

Select the **OPC UA** menu item in the **Network/Remote Access** group.

The **OPC UA NC Server** area contains the following functions:

Function	Meaning
Status	Shows with an icon whether a connection with the OPC UA NC Server is active: ■ Green icon: connection active ■ Gray icon: connection not active or software option not enabled
OPC UA connection wizard	Open the **OPC UA NC Server connection wizard** window **Further information:** "OPC UA connection wizard function (options 56 to 61)", Page 1934
OPC UA license settings	Open the **OPC UA NC Server - License Settings** window **Further information:** "OPC UA license settings function (options 56 to 61)", Page 1935
Host computer operation	Activate or deactivate host computer operation with a switch **Further information:** "DNC area", Page 1936

38.12.3 OPC UA connection wizard function (options 56 to 61)

Application

For quick and easy setup of an OPC UA client application, you can use the **OPC UA NC Server connection wizard** window. This assistant guides you through the steps that are required to connect an OPC UA client application to the control.

Related topics

- Assigning an OPC UA client to a software option 56 to 61 with the **OPC UA NC Server - License Settings** window
- Managing certificates with the **PKI Admin** menu

Description of function

The **OPC UA connection wizard** function of the **OPC UA** menu item opens the **OPC UA NC Server connection wizard** window.

Further information: "OPC UA menu item (options 56 to 61)", Page 1934

The assistant features the following steps:

- Export **OPC UA NC Server** certificates
- Import the certificates of the OPC UA client application
- Assign each of the available **OPC UA NC Server** software options to an OPC UA client application
- Import user certificates
- Assign user certificates to users
- Configure the firewall

If at least one of the options 56 to 61 is active, then, when booting for the first time, the control creates the server certificate as part of a self-generated certificate chain. The client application or the manufacturer of the application creates the client certificate. The user certificate is linked to the user account. Please contact your IT department.

Notes

- The **OPC UA NC Server connection wizard** also helps you create test or sample certificates for users and the OPC UA client application. Do not use the user and client application certificates created at the control for other purposes than development at the programming station.

38.12.4 OPC UA license settings function (options 56 to 61)

Application

Use the **OPC UA NC Server - License Settings** window to assign an OPC UA client-application to one of the software options 56 to 61.

Related topics

- Setting up an OPC UA client application with the **OPC UA connection wizard** function

  Further information: "OPC UA connection wizard function (options 56 to 61)", Page 1934

Description of function

After using the **OPC UA connection wizard** or the **PKI Admin** menu item to import a certificate of an OPC UA client application, you can choose the certificate from a selection window.

If you activate the **Active** check box for a certificate, the control uses a software option for the OPC UA client application.

38.13 DNC menu item

Application

With the **DNC** menu item you can grant or restrict access to the control (e.g., connections over a network).

Related topics

- Connecting network drives
  Further information: "Network drives on the control", Page 1922
- Setting up a network
  Further information: "Ethernet interface", Page 1925
- TNCremo
  Further information: "PC software for data transfer", Page 1970
- Remote Desktop Manager (option 133)
  Further information: "Remote Desktop Manager window (option 133)", Page 1944

Description of function

The **DNC** area contains the following icons:

Icon	Meaning
	External access to the control is active
	Add computer-specific connection
	Edit computer-specific connection
	Delete computer-specific connection

DNC area

In the **DNC** area you use switches to activate the following functions:

Switches	Meaning
DNC access permitted	Permit or block all accesses to the control through a network or a serial connection
TNCopt full access allowed	Depending on the machine, permit or block access for diagnostics or initial setup software
Host computer operation	Pass command control to an external host computer, for example to transfer data to the control; or end host computer operation If host computer operation is active, the control displays the **Host computer is active** message in the info bar. You cannot use the **Manual** and **Program Run** operating modes. You cannot activate host computer operation while running an NC program.

Computer-specific connections

If the machine manufacturer has defined the optional machine parameter **CfgAccessControl** (no. 123400), then in the **Connections** area you can permit or block access for up to 32 connections defined by you.

The control shows the defined information in a table:

Column	Meaning
Name	Host name of the external computer
Description	Additional information
IP address	Network address of the external computer
Access	■ **Permit** The control permits network access without confirmation. ■ **Inquire** The control asks for confirmation upon a network access attempt. You can choose whether to permit or block the access once or always. ■ **Deny** The control does not permit any network access
Type	■ **Com1** Serial interface 1 ■ **Com2** Serial interface 2 ■ **Ethernet** Network connection
Active	If a connection is active, the control displays a green circle. If a connection is inactive, the control displays a gray circle.

Notes

- In the machine parameter **allowDisable** (no. 129202) the machine manufacturer defines whether the **Host computer operation** switch is available.
- In the optional machine parameter **denyAllConnections** (no. 123403) the machine manufacturer defines whether the control permits computer-specific connections.

38.14 Printers

Application

You add and manage printers through the **Printer** menu item in the **Heros Printer Manager** window.

Related topics

- Using the **FN 16: F-PRINT** function for printing

  Further information: "Outputting text formatted with FN 16: F-PRINT", Page 1285

Requirement

- PostScript-capable printer

  The control can communicate only with printers that understand PostScript emulation such as KPDL3. Some printers enable setting the PostScript emulation in the printer menu.

  Further information: "Note", Page 1940

Description of function

The **Printer** menu item opens the **Heros Printer Manager** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

You can print the following files:

- Text files
- Graphic files
- PDF files

Further information: "File types", Page 1071

Once you have added a printer, the control shows the **PRINTER:** drive in the file management. The drive contains one folder for each defined printer.

Further information: "Creating a printer", Page 1940

There are various methods to start printing:

- Copying the file to be printed to the **PRINTER** drive

  The file to be printed is automatically forwarded to the default printer and deleted from the directory after the print job has been executed.

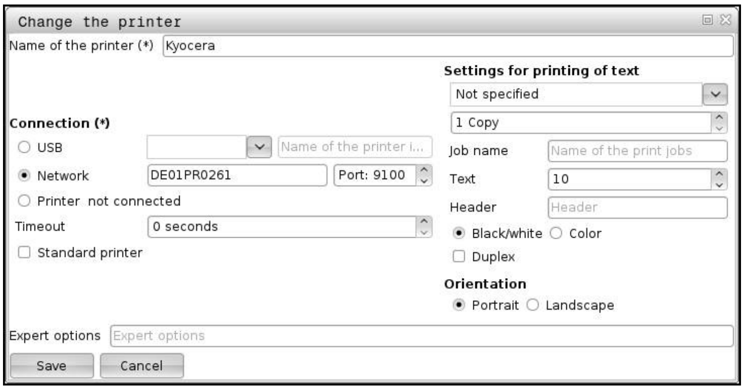
  You may also copy the file into the printer sub-directory if you wish to use a printer other than the default printer.
- Using the **FN 16: F-PRINT** function

Buttons

The **Heros Printer Manager** window contains the following buttons:

Button	Meaning
Create	Creates a printer
CHANGE	Adapts the properties of the selected printer
COPY	Creates a copy of the selected printer setting At first the copy has the same properties as the copied setting. This can be useful if printing both portrait and landscape formats on the same printer
DELETE	Deletes the selected printer
UP **DOWN**	Selects a printer
STATUS	Displays the status information of the selected printer
PRINT A TEST PAGE	Prints a test page on the selected printer

Change the printer window



For each printer, the following properties can be set:

Setting	Meaning
Name of the printer	Customizes the printer name
Connection	Selects the connection ■ **USB**: The control automatically displays the name ■ **Network**: Network name or IP address of the printer Port for the network printer (default: 9001) ■ **Printer %1 not connected**
Timeout	Delays the printing process The control delays the printing process by the pre-set number of seconds after the last change has been made to the file to be printed in **PRINTER:**. Use this setting if the file to be printed is populated with FN functions, e.g. when probing.
Standard printer	Selects the default printer The control automatically assigns this setting to the first printer added.
Settings for printing of text	These settings are applicable when printing text documents: ■ Paper size ■ Number of copies ■ Job name ■ Font size ■ Header ■ Print options (black and white, color, duplex)
Orientation	Portrait or landscape for all printable files
Expert options	Available only to authorized specialists

38.14.1 Creating a printer

To create a new printer:

- ▶ Enter the printer name in the name dialog
- ▶ Select **Create**
- > The control creates a new printer.
- ▶ Press **CHANGE**
- > The control opens the **Change the printer** window.
- ▶ Define the properties
- ▶ Select **Save**
- > The control applies the settings and displays the defined printer in the list.

Note

If your printer does not permit PostScript emulation, change the printer settings if possible.

38.15 VNC menu item

Application

VNC is software that shows the screen contents of a remote computer on a local computer, and also sends keyboard actions and mouse movements of the local computer to the remote computer.

Related topics

- Firewall settings

  Further information: "Firewall", Page 1950
- Remote Desktop Manager (option 133)

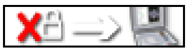
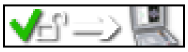
  Further information: "Remote Desktop Manager window (option 133)", Page 1944

Description of function

The **VNC** menu item opens the **VNC settings** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

Buttons and icons

The **VNC settings** window contains the following buttons and icons:

Button and icon	Meaning
Add	Add new VNC viewer or client
Remove	Delete the selected client Only possible with manually entered clients.
Edit	Edit the configuration of the selected client
Update	Refresh view Required with connection attempts during which the dialog is open.
Set preferred owner of the focus	Activate the **Preferred owner of the focus** check box
	Another client owns the focus Mouse and keyboard are disabled
	You own the focus Entries can be made
	Prompt by another client to receive the focus Mouse and keyboard are disabled until the focus is assigned.

VNC participant settings area

In the **VNC participant settings** area, the control shows a list of all clients.

The control displays the following contents:

Column	Contents
Computer name	IP address or computer name
VNC	Connection of the client to the VNC viewer
VNC Focus	The client participates in the focus assignment
Type	■ Manual Manually entered client ■ Denied This client is not permitted to connect. ■ Enable TeleService and IPC Client via a TeleService connection ■ DHCP Other computer that retrieves an IP address from this computer.

Global settings area

In the **Global settings** area you can define the following settings:

Function	Meaning
Enable TeleService and IPC	If the check box is selected, the connection is always permitted.
Password verification	Client must enter a password for verification The control opens a window when you select the check box. In this window you define the password for this client. The client must enter the password when establishing the connection.

Enabling other VNC area

In the **Enabling other VNC** area you can define the following settings:

Function	Meaning
Deny	Other VNC clients are not permitted.
Inquire	A dialog opens when another VNC client wants to connect. You must grant permission for this connection.
Permitted	Other VNC clients are permitted.

VNC Focus Settings area

In the **VNC Focus Settings** area, you can make the following settings:

Function	Meaning
Enabling VNC focus	Enables focus assignment for this system When the check box is inactive, the focus owner actively gives away the focus by using the focus symbol. The remaining clients can request the focus only after it was given away.
Reset the CapsLock key when changing the focus	When the check box is active and the focus owner has activated the CapsLock key, the CapsLock key is deactivated in a focus change. Only if the **Enabling VNC focus** check box is active
Enable Concurrency VNC Focus	When the check box is active, every client can request the focus at any time. The focus owner does not need to give away the focus before to enable that. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the **Enabling VNC focus** check box is active
Timeout Concurrency VNC Focus	Period of time after requesting the focus during which the focus owner can object to the focus change (at most 60 seconds). This period of time is set by moving a slider. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the **Enabling VNC focus** check box is active

Activate the **Enabling VNC focus** check box only in connection with HEIDENHAIN devices provided especially for this purpose, e.g. ITC industrial computers.

Notes

- The machine manufacturer defines the procedure for assigning the focus with multiple clients or operating units. Focus assignment depends on the setup and operating situation of the machine tool.

  Refer to your machine manual.
- The control displays a message if the firewall settings of the control do not permit the VNC protocol for all clients.

Definition

Abbreviation	Definition
VNC (virtual network computing)	**VNC** is software with which another computer can be controlled over a network connection.

38.16 Remote Desktop Manager window (option 133)

Application

With Remote Desktop Manager you can display external computer units on the control screen that are connected via Ethernet, and operate them through the control. You can also shut down a Windows computer together with the control.

Related topics

- External access

  Further information: "DNC menu item", Page 1936

Requirement

- Remote Desktop Manager (software option 133)
- Existing network connection

  Further information: "Ethernet interface", Page 1925

Description of function

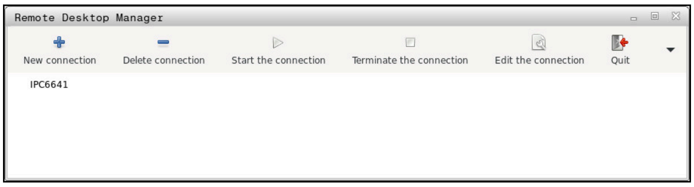
The **Remote Desktop Manager** menu item opens the **Remote Desktop Manager** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

Remote Desktop Manager grants the following connection options:

- **Windows Terminal Service (RemoteFX)**: Display the desktop of an external Windows computer on the control

  Further information: "Windows Terminal Service (RemoteFX)", Page 1945
- **VNC**: Display the desktop of an external Windows, Apple or Unix computer on the control

  Further information: "VNC", Page 1945
- **Switch-off/restart of a computer**: Automatically shut down a Windows computer together with the control
- **World Wide Web**: Only for authorized specialists
- **SSH**: Only for authorized specialists
- **XDMCP**: Only for authorized specialists
- **User-defined connection**: Only for authorized specialists



HEIDENHAIN offers the IPC 6641 as a Windows computer. With the IPC 6641 you can start and operate Windows-based applications directly from within the control.

If the desktop of the external connection or the external computer is active, all inputs from the mouse and the alphabetic keyboard are transmitted there.

When the operating system is shut down, the control automatically terminates all connections. Please note that only the connection is terminated, whereas the external computer or the external system is not shut down automatically.

Buttons

Remote Desktop Manager contains the following buttons:

Button	Function
New connection	Use the **Edit the connection** window to create a new connection **Further information:** "Establishing and starting a connection", Page 1949
Delete connection	Delete the selected connection
Start the connection	Start the selected connection **Further information:** "Establishing and starting a connection", Page 1949
Terminate the connection	Terminate the selected connection
Edit the connection	Use the **Edit the connection** window to modify the selected connection **Further information:** "Connection settings", Page 1946
Exit	Close **Remote Desktop Manager**
Import connections	Restore the selected connection **Further information:** "Exporting and importing connections", Page 1949
Export the connections	Back-up the selected connection **Further information:** "Exporting and importing connections", Page 1949

Windows Terminal Service (RemoteFX)

You don't need any additional software on a computer for a RemoteFX connection, but you might need to change some settings on the computer.

Further information: "Configuring an external computer for Windows Terminal Service (RemoteFX)", Page 1948

For integrating the IPC 6641, HEIDENHAIN recommends using a RemoteFX connection.

With RemoteFX, a separate window opens for the screen of the external computer. The active desktop on the external computer is then locked and the user logged off. This prevents two users from accessing the control simultaneously.

VNC

You need an additional **VNC** server for your external computer when connecting through VNC. Install and configure the VNC server, e.g. TightVNC server, before establishing the connection.

VNC mirrors the screen of the external computer. The active desktop on the external computer is not locked automatically.

With a **VNC** connection you can shut down the external computer through the Windows menu. The computer cannot be restarted through the connection.

Connection settings

General settings

The following settings apply to all connection options:

Setting	Meaning	Intended use
Connection name	Name of the connection in **Remote Desktop Manager** You can use the following characters in the name of the connection: A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _	Required
Restarting after end of connection	Behavior after disconnection: ■ **Always restart** ■ **Never restart** ■ **Always after an error** ■ **Ask after an error**	Required
Automatic starting upon login	Connect automatically when starting	Required
Add to favorites	The control displays the connection's icon in the taskbar. Tap or click the icon to start the connection directly.	Required
Move to the following workspace	Number of the desktop for the connection; desktops 0 and 1 are reserved for the NC software. Default setting: Third desktop	Required
Release USB mass memory	Permit access to connected USB mass memory devices	Required
Private connection	Connection can be seen and used only by its creator	Required
Computer	Host name or IP address of the external computer HEIDENHAIN recommends the **IPC6641.machine.net** setting for the IPC 6641. The host name **IPC6641** must be assigned to the IPC in the Windows operating system for this setting.	Required
Password	Password of the user	Required
Entries in the **Advanced options** area	Available only to authorized specialists	Optional

Additional settings for Windows Terminal Service (RemoteFX)

The control offers the following additional connection settings for the **Windows Terminal Service (RemoteFX)** option:

Setting	Meaning	Intended use
User name	Name of the user	Required
Windows domain	Domain of the external computer	Optional
Full-screen mode or **User-defined window size**	Size of the connection window on the control	Required

Additional settings for VNC

The control offers the following additional connection settings for the **VNC** option:

Setting	Meaning	Intended use
Full-screen mode or **User-defined window size:**	Size of the connection window on the control	Required
Permit further connections (share)	Additionally grant other VNC connections access to the VNC server	Required
View only	In display mode, the external computer cannot be operated.	Required

Additional settings for Switch-off/restart of a computer

The control offers the following additional connection settings for the **Switch-off/restart of a computer** option:

Setting	Meaning	Intended use
User name	User name with which the connection should log in.	Required
Windows domain:	If required, domain of the target computer	Optional
Max. waiting time (seconds):	A shutdown of the control causes the Windows computer to shut down as well. Before the control displays the **Now you can switch off.** message, it waits for the number of seconds defined here. While waiting, the control checks whether the Windows computer is still accessible (port 445). If the Windows computer is switched off before the defined number of seconds have expired, the control will wait no longer.	Required
Additional waiting time:	Waiting time after the Windows computer has stopped being accessible. Windows applications may delay the shutdown of the computer after port 445 has been closed.	Required
Force	Close all programs on the Windows computer, even if dialogs are still open. If **Force** is not selected, Windows waits up to 20 seconds. This delays the shutdown process or the Windows computer is switched off before Windows has shut down.	Required
Restart	Restart the windows computer	Required
Run during restart	When the control restarts, restart the Windows computer as well. Effective only if the control is restarted using the shutdown icon at the bottom right in the taskbar or if it is restarted as a result of a change in the system settings (e.g. network settings).	Required
Run during switch-off	Shut down the Windows computer (no restart) when shutting down the control. This is the default behavior. Even the **END** key will then not trigger a restart.	Required

38.16.1 Configuring an external computer for Windows Terminal Service (RemoteFX)

To configure the external computer (e.g., in Windows 10 operating systems):

- Press the Windows key
- Select **Control Panel**
- Select **System and Security**
- Select **System**
- Select **Remote Settings**
- > The computer opens a pop-up window.
- Under **Remote Assistance**, enable **Allow Remote Assistance connections to this computer**
- In the **Remote Desktop** area, enable **Allow Remote Assistance connections to this computer**
- Press **OK** to confirm your settings

38.16.2 Establishing and starting a connection

To establish and start a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select **New connection**
- > The control displays a selection menu.
- ▶ Select a connection option
- ▶ Under **Windows Terminal Service (RemoteFX)**, select the operating system
- > The control opens the **Edit the connection** window.
- ▶ Define the connection settings

  Further information: "Connection settings", Page 1946
- ▶ Press **OK**
- > The control saves the settings and closes the window.
- ▶ Select connection
- ▶ Select **Start the connection**
- > The control starts the connection.

38.16.3 Exporting and importing connections

To export a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the desired connection
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Export the connections**
- > The control opens the **Select export file** window.
- ▶ Define the name of the saved file
- ▶ Select the target file
- ▶ Select **Save**
- > The control saves the connection data under the name defined in the window.

To import a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Import connections**
- > The control opens the **Select file to import** window.
- ▶ Select file
- ▶ Select **Open**
- > The control creates the connection under the name that was defined originally in the **Remote Desktop Manager**.

Notes

NOTICE

Caution: Data may be lost!

If you do not shut down external computers properly, data may be irreversibly damaged or deleted.

- Configure the automatic shutdown of the Windows computer

- When you edit an existing connection, the control will automatically delete all impermissible characters from the name.

Notes in connection with the IPC 6641

- HEIDENHAIN assures a functioning connection between HEROS 5 and the IPC 6641. No guarantee is given for other combinations and connections.
- If you use the computer name **IPC6641.machine.net** to connect an IPC 6641, it is important to enter **.machine.net**.

  With this entry, the control automatically searches the Ethernet interface **X116**, and not the interface **X26**; this reduces the time needed for access.

38.17 Firewall

Application

With the control you can set up a firewall for the primary network interface, and for a sandbox if needed. You can block incoming network traffic for specific senders and services.

Related topics

- Existing network connection

  Further information: "Ethernet interface", Page 1925
- SELinux security software

  Further information: "SELinux security software", Page 1921

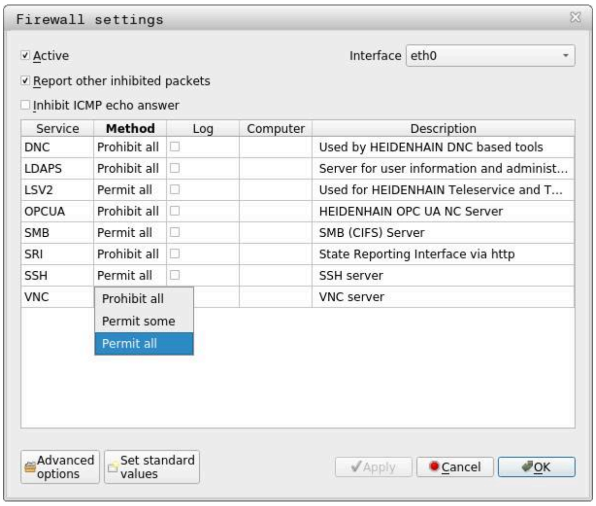
Description of function

The **Firewall** menu item opens the **Firewall settings** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

If you activate the firewall, the control displays an icon at the bottom right in the taskbar. The control displays the following icons, depending on the security level:

Icon	Meaning
	Firewall protection does not yet exist although it has been activated. Example: A dynamic IP address is used in the network inter-face configuration, but the DHCP server has not yet assigned an IP address. **Further information:** "DHCP server tab", Page 1930
	Firewall active with medium security level.
	Firewall active with high security level. All services except for SSH are blocked.

Firewall settings



The **Firewall settings** window contains the following settings:

Setting	Meaning
Active	Activate or deactivate firewall
Interface	Select the interface ■ **eth0**: X26 of the control ■ **eth1**: X116 of the control ■ **brsb0**: Sandbox (optional) If a control has two Ethernet interfaces, then by default the DHCP server for the machine network is active for the second interface. With this setting you cannot activate the firewall for **eth1** because the firewall and DHCP server mutually exclude each other.
Report other inhibit-ed packets	Activate the firewall with a high security level All services except for SSH are blocked.
Inhibit ICMP echo answer	If this check box is selected, the control does not respond to a ping request.

<table>
<tr><th>Setting</th><th>Meaning</th></tr>
<tr><td>Service</td><td>Brief designation of services configured with the firewall. You can change the settings even if the services are not started.
DNC
DNC server using the RPC protocol for external applications that were developed with RemoTools SDK (port 19003)
For more detailed information, consult the RemoTools SDK manual.
LDAPS
Server with user data and configuration of user administration
LSV2
Functionality for TNCremo, TeleService, and other HEIDENHAIN PC tools (port 19000)
OPC UA
Service provided by the OPC UA NC Server (port 4840).
SMB
Only incoming SMB connections, meaning a Windows share on the control. Outgoing SMB connections are not influenced, meaning a Windows share connected to the control.
SSH
SecureShell protocol (port 22) for secure LSV2 handling with active user administration; starting with HEROS 504
VNC
Access to screen contents. If you block this service, then not even TeleService programs from HEIDENHAIN can access the control. If you block this service, the control displays a warning in the VNC settings window.
Further information: "VNC menu item", Page 1940</td></tr>
<tr><td>Method</td><td>Configure accessibility
Prohibit all: Cannot be accessed by anyone
Permit all: Can be accessed by everyone
Permit some: Can be accessed only by specific clients
In the Computer column you must define the computer for which access is permitted. If you do not define a computer, the control activates Prohibit all.</td></tr>
<tr><td>Log</td><td>The control shows the following messages when transmitting network packets:
Red: Network packet blocked
Blue: Network packet accepted</td></tr>
<tr><td>Computer</td><td>IP address or host name of the computers with access rights. Separated by commas, if there are multiple computers
The control converts the host name to an IP address when the control starts. If the IP address changes, you must restart the control or change the setting. The control issues an error message if it cannot convert the host name to an IP address.
Only with the Permit some method</td></tr>
<tr><td>Advanced options</td><td>Only for network specialists</td></tr>
<tr><td>Set standard values</td><td>Reset the settings to the default values recommended by HEIDENHAIN</td></tr>
</table>

Notes

- Have your network specialist check and, if necessary, change the standard settings.
- The firewall does not protect the second network interface **eth1**. Connect only trustworthy hardware to this interface, and do not use this interface for Internet connections.

38.18 Portscan

Application

With the **Portscan** function, the control checks all open, incoming TCP and UDP listen ports at defined intervals or when commanded. The control shows a message if a port is not listed.

Related topics

- Firewall settings

  Further information: "Firewall", Page 1950
- Network settings

  Further information: "Network settings window", Page 1927

Description of function

The **Portscan** menu item opens the **Portscan** window. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

The control searches for all open, incoming TCP and UDP listen ports on the system and compares them to the following whitelists:

- System-internal whitelists **/etc/sysconfig/portscan-whitelist.cfg** and **/mnt/sys/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with machine-manufacturer-specific functions: **/mnt/plc/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with customer-specific functions: **/mnt/tnc/etc/sysconfig/portscan-whitelist.cfg**

Each whitelist contains the following information:

- Port type (TCP/UDP)
- Port number
- Offering program
- Comments (optional)

In the **Manual Execution** area, use the **Start** button to start the portscan manually. Use the **Automatic update on** function of the **Automatic Execution** to have the control automatically perform the portscan at a defined interval. You define the interval with a slider.

If the control performs the portscan automatically, then only ports listed in the whitelists my be open. The control shows a message window if a port is not listed.

38.19 Remote servicing

Application

In conjunction with the remote service setup tool, TeleService from HEIDENHAIN offers the possibility to establish encrypted end-to-end connections between a computer and a machine over the Internet.

Related topics

- External access

  Further information: "DNC menu item", Page 1936
- Firewall

  Further information: "Firewall", Page 1950

Requirements

- Existing Internet connection

  Further information: "Network settings window", Page 1927
- **LSV2** connection permitted by the firewall

  Remote diagnosis via the TeleService PC software uses the **LSV2** service. By default, the control's firewall blocks all incoming and outgoing connections. For this reason you must permit a connection for this service.

  You have the following options for permitting the connection:

  - Deactivate the firewall
  - Define the **Permit some** method for the **LSV2** service, and enter the name of the computer under **Computer**

  Further information: "Firewall", Page 1950

Description of function

The **Remote Service** menu item opens the **HEIDENHAIN remote maintenance** window. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

You need a valid session certificate for the servicing session.

Session certificate

During installation of the NC software, a temporary certificate is automatically installed on the control. A new installation or update may only be carried out by a service technician from the machine manufacturer.

A new certificate must be installed if no valid session certificate is installed on the control. Clarify with the service technician which certificate is needed. The service technician will then provide you with a valid certificate file, if necessary, which you must then install.

Further information: "Installing a session certificate", Page 1955

In order to start the servicing session, you must enter the session key from the machine manufacturer.

38.19.1 Installing a session certificate

To install a session certificate on the control:

- Select the **Settings** application
- Select **Network/Remote Access**
- Double-tap or double-click **Network**
- > The control opens the **Network settings** window.
- Select the **Internet** tab

> The machine manufacturer defines the settings in the **Telemaintenance** field.

- Select **Add**
- > The control displays a selection menu.
- Select file
- Select **Open**
- > The control opens the certificate.
- Select **OK**
- Restart the control to load the settings

Notes

- If you've deactivated the firewall, you must reactivate it after the servicing session!
- If you permit the **LSV2** service in the firewall, the access security is ensured through the network settings. The network security is the responsibility of the machine manufacturer or the respective network administrator.

38.20 Backup and restore

Application

With the **NC/PLC Backup** and **NC/PLC Restore** functions you can back up and restore individual folders or the entire **TNC:** drive. You can save the backup files to various types of memory media.

Related topics

- File management, **TNC:** drive

  Further information: "File management", Page 1066

Description of function

You open the backup function through the **NC/PLC Backup** menu item. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

You open the restore function through the **NC/PLC Backup** menu item.

The backup function creates a ***.tncbck** file. The restore function can restore these files as well as files from existing TNCbackup programs. If you double tap or click a ***.tncbck** file in the file manager, the control starts the restore function.

Further information: "File management", Page 1066

Within the backup function you can chose between the following types of backups:

- **Back up the "TNC:" partition**

  Back-up all data on the **TNC:** drive
- **Back up the directory tree**

  Back-up the selected folders and their subfolders on the **TNC:** drive
- **Back up the machine configuration**

  Only for the machine manufacturer
- **Complete backup (TNC: and machine configuration)**

  Only for the machine manufacturer

Backup and restore is subdivided into several steps. Navigate between these steps with the **FORWARD** and **BACK** buttons.

38.20.1 Backing up data

To back-up the data of the **TNC:** drive:

- Select the **Settings** application
- Select **Diagnostics/Maintenance**
- Double-tap or double-click **NC/PLC Backup**
- > The control opens the **Back up the "TNC:" partition** window.
- Specify the type of backup
- Select **Forward**
- If necessary, pause the control with **Stop NC software**
- Select any predefined exclusion rules or ones you have defined yourself
- Select **Forward**
- > The control generates a list of files for backing up.
- Check list
- Deselect files if necessary
- Select **Forward**
- Enter the name of the backup file
- Select the storage path
- Select **Forward**
- > The control generates the backup file.
- Confirm with **OK**
- > The control concludes the backup process and restarts the NC software.

38.20.2 Restoring data

NOTICE

Caution: Data may be lost!

When you restore data (Restore function), any existing data will be overwritten without a confirmation prompt. Existing data is not automatically backed up by the control before running the restore process. Power failures or other problems can interfere with the data restore process. As a consequence, data may be irreversibly damaged or deleted.

- Before starting the data restore process, make a backup of the existing data

To restore data:

- Select the **Settings** application
- Select **Diagnostics/Maintenance**
- Double-tap or double-click **NC/PLC Restore**
- > The control opens the **Restore data - %1** window.
- Select the archive to be restored
- Select **Forward**
- > The control generates a list of files for restoring.
- Check list
- Deselect files if necessary
- Select **Forward**
- If necessary, pause the control with **Stop NC software**
- Select **Extract archive**
- > The control restores the files.
- Confirm with **OK**
- > The control restarts the NC software.

Note

The TNCbackup PC program can also process ***.tncbck** files. TNCbackup is part of TNCremo.

38.21 TNCdiag

Application

The control displays status and diagnostic information of HEIDENHAIN components in the **TNCdiag** window.

Description of function

Only use this function after consultation with your machine manufacturer.

For more information, please refer to the **TNCdiag** documentation.

38.22 Machine parameters

Application

You can configure the behavior of the control with machine parameters. The control offers the applications **MPs for users** and **MPs for setters** for this. You can open the **MPs for users** application at any time without having to enter a code number.

The machine manufacturer defines which machine parameters are in which applications. HEIDENHAIN offers a standard scope of parameters for the **MPs for setters** application. The following contents describe only the standard scope of the **MPs for setters** application.

Related topics

- List of machine parameters in the **MPs for setters** application

  Further information: "Machine parameters", Page 1980

Requirements

- Code number 123

  Further information: "Code numbers", Page 1914
- Contents of the **MPs for setters** application are defined by the machine manufacturer

Description of function

The **MPs for setters** menu item opens the **MPs for setters** application. The menu item is in the **Machine parameters** group of the **Settings** application.

In the **Machine parameters** group the control shows only those menu items that you can choose with the current access rights.

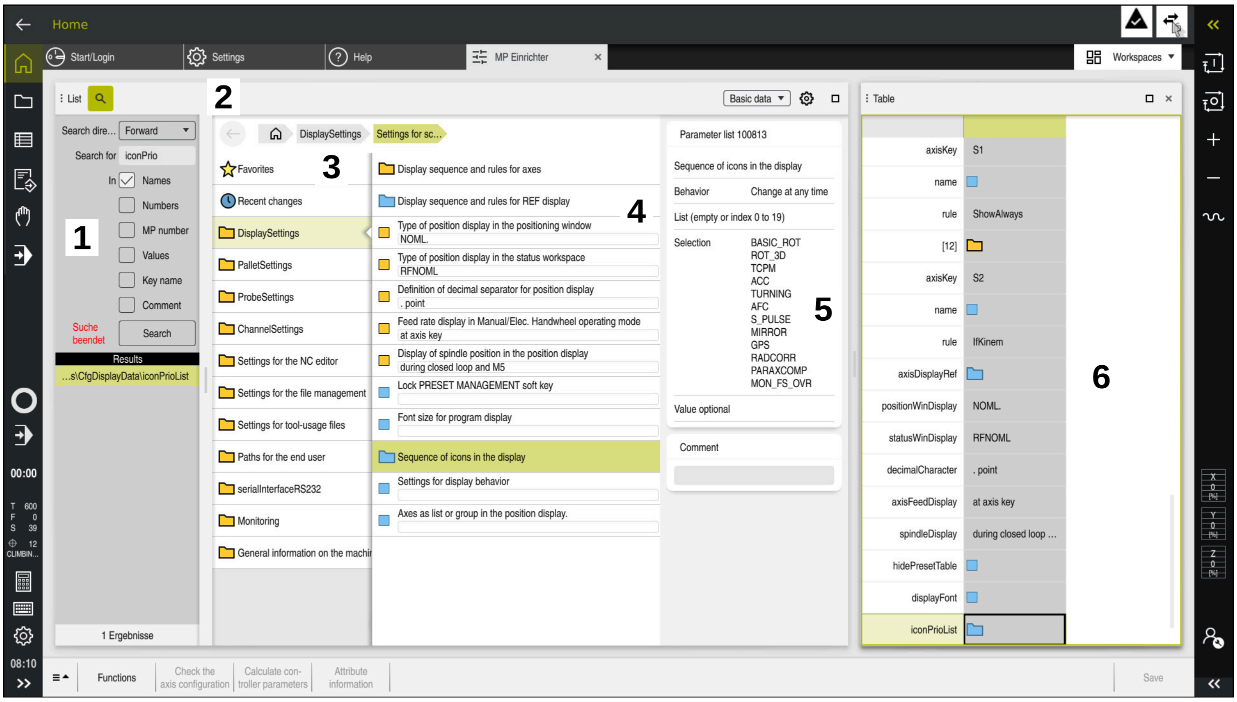
If you open an application for machine parameters, the control displays the configuration editor.

The configuration editor offers the following workspaces:

- **List**
- **Table**

You can't close the **List** workspace.

Areas of the configuration editor



MPs for setters application with selected machine parameter

The configuration editor shows the following areas:

1 **Search** column

You can search forward or backward with the following characteristics:

- Name

  This is the language-neutral name used for machine parameters in the User's Manual.
- Number

  This is the unique number used for machine parameters in the User's Manual.
- MP number of the iTNC 530
- Value
- Key name

  Machine parameters for axes or channels exist more than once. In order to avoid ambiguity, each axis and each channel is identified with a key name (e.g., **X1**)
- Comment

The control displays the results.

2 Title bar of the **List** workspace

You can show and hide the **Search** column, use a selection menu to filter the contents, and open the **Configuration** window.

Further information: "Configuration window", Page 1962

3 Navigation column

The control provides the following options for navigation:

- Navigation path
- Favorites
- 21 most recent changes
- Structure of the machine parameters

4 Content column

In the content column the control displays objects, machine parameters, or changes that you select using the search function or navigation column.

5 Information area

The control displays information about the selected machine parameter or change.

Further information: "Information area", Page 1962

6 **Table** workspace

In the **Table** workspace the control shows the selected contents within the structure. In order to do so, in the **Configuration** window the **Synchronized navigation in list and table** switch must be set to active.

The control displays the following information:

- Name of the objects
- Icon of the objects
- Value of the machine parameters

Icons and buttons

The configuration editor contains the following icons and buttons:

Icon or button	Meaning
	Open the **Configuration** window **Further information:** "Configuration window", Page 1962
	Select **Recent changes**
	Object exists ■ Data object ■ Directory ■ Parameter list
	Object empty
	Machine parameter exists
	Optional machine parameter does not exist
	Machine parameter invalid
	Machine parameter readable but not editable
	Machine parameter unreadable and uneditable
	Changes to the machine parameter not yet saved
Functions	Open the context menu **Further information:** "Context menu", Page 1392
Check the axis configuration	Only for the machine manufacturer
Calculate controller parameters	Only for the machine manufacturer
Attribute information	Only for the machine manufacturer
Save	The control opens a window with all of the changes since the most recent saving. You can save or discard the changes.

Configuration window

In the **Configuration** window you define the settings for displaying the machine parameters in the configuration editor.

The **Configuration** window consists of the following areas:

- **List**
- **Table**

The **List** area contains the following settings:

Setting	Meaning
Show MP descriptive texts	If the switch is active, the control displays a description of the machine parameter in the active conversational language. If the switch is not active, the control displays the language-neutral name of the machine parameter.
Show details	Hide or show the information area with this switch.

The **Table** area contains the following settings:

Setting	Meaning
Show details when table displayed	If the switch is active, the control shows the information area even if the **Table** workspace is open. If the switch is not active, the control shows the information area only if the **Table** workspace is closed.
Synchronized navigation in list and table	If the switch is active, the control always shows in the **Table** workspace the object that is marked in the **List** workspace, and vice versa. If the switch is not active, the contents of the two workspaces do not synchro-nize.

Information area

If you select contents from the favorites or the structure, the control will display some of the information below in the information area:

- Type of object, such as data object list or parameter, perhaps with number
- Descriptive text of machine parameter
- Information about the effect
- Permitted or required input
- Behavior, such as program run disabled
- MP number of the iTNC 530 for machine parameters
- Machine parameter optional

If you select content from one of the recent changes, the control will display the information below in the information area:

- Sequential number of the change
- Previous value
- New value
- Date and time of change
- Descriptive text of machine parameter
- Information about the effect

39

HEROS Operating System

39.1 Fundamentals

HEROS is the fundamental basis for all NC controls from HEIDENHAIN. The HEROS operating system is based on Linux, and was adapted for the purposes of NC controls.

The TNC7 features the version HEROS 5.

39.2 HEROS menu

Application

In the HEROS menu the control shows information about the operating system. You can change settings or use HEROS functions.

By default you open the HEROS menu through the taskbar at the bottom edge of the screen

Related topics

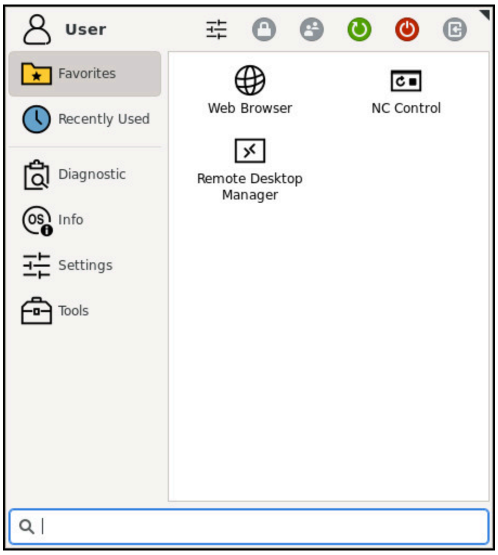
- Opening HEROS functions through the **Settings** application

  Further information: "Settings Application", Page 1911

Description of function

You open the HEROS menu with the green DIADUR icon in the taskbar or with the **DIADUR** key.

Further information: "Taskbar", Page 1967



Standard view of the HEROS menu

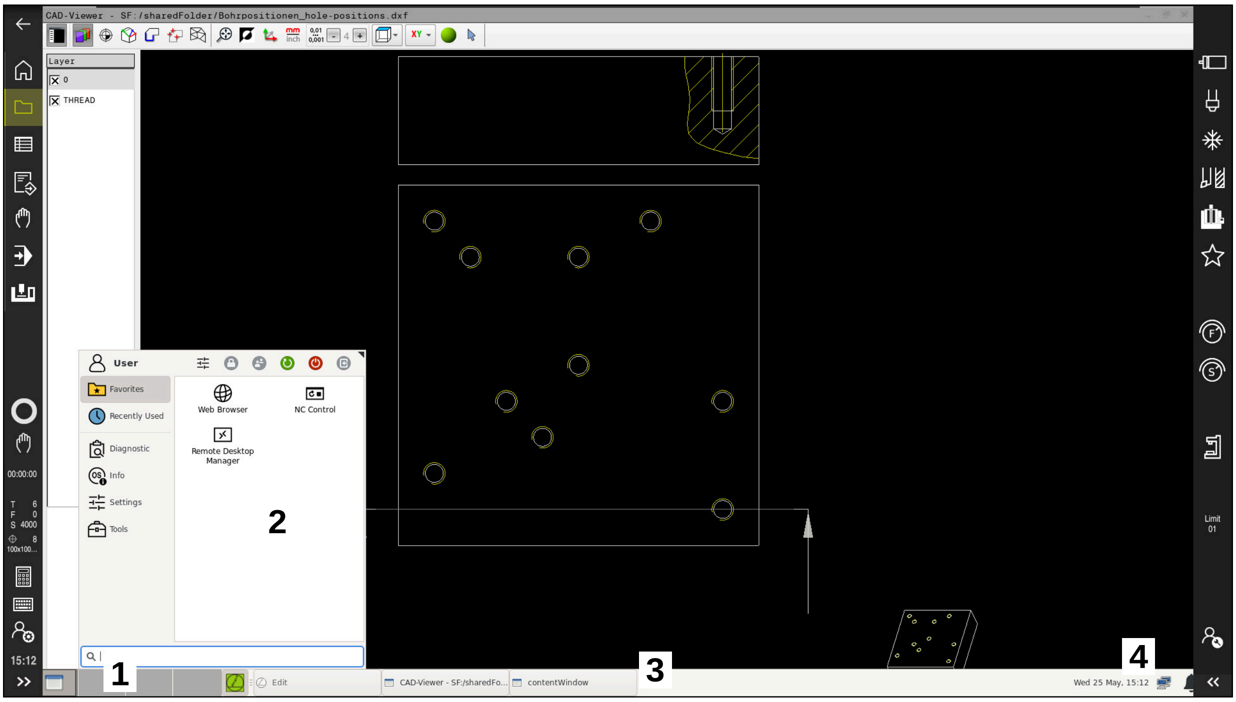
The HEROS menu contains the following functions:

Area	Function
Header	■ User name: Currently no function ■ User-specific settings ■ Lock display ■ Switch user: Currently no function ■ Restart ■ Shut down ■ Log out: Currently no function

Area	Function
Navigation	■ Favorites ■ Recently used
Diagnostic	■ **GSmartControl**: Available only to authorized specialists ■ **HE Logging**: Define settings for internal diagnostic files ■ **HeMenu**: Available only to authorized specialists ■ **perf2**: Check processor load and process load ■ **Portscan**: Test active connections **Further information:** "Portscan", Page 1953 ■ **Portscan OEM**: Available only to authorized specialists ■ **RemoteService**: Start and stop remote maintenance **Further information:** "Remote servicing", Page 1954 ■ **Terminal**: Enter and execute console commands ■ **TNCdiag**: Evaluates status and diagnostic information of HEIDENHAIN components with a focus on the drives and presents it graphically **Further information:** "TNCdiag", Page 1957
Settings	■ **screensaver** ■ **Current User**: Currently no function ■ **Date/Time** **Further information:** "Adjust system time window", Page 1919 ■ **Firewall** **Further information:** "Firewall", Page 1950 ■ **HePacketManager**: Available only to authorized specialists ■ **HePacketManager Custom**: Available only to authorized specialists ■ **Language/Keyboards** **Further information:** "Conversational language of the control", Page 1919 ■ **License Settings** **Further information:** "OPC UA NC Server (options 56 to 61)", Page 1931 ■ **Network** **Further information:** "Ethernet interface", Page 1925 ■ **OEM Function Users**: Currently no function ■ **One Click Setup**: Currently no function ■ **OPC UA / PKI Admin** **Further information:** "OPC UA NC Server (options 56 to 61)", Page 1931 ■ **Printer** **Further information:** "Printers", Page 1937 ■ **SELinux** **Further information:** "SELinux security software", Page 1921 ■ **Shares** **Further information:** "Network drives on the control", Page 1922 ■ **UserAdmin**: Currently no function ■ **VNC** **Further information:** "VNC menu item", Page 1940 ■ **WindowManagerConfig**: Settings for the Window Manager **Further information:** "Window Manager", Page 1968

Area	Function
Info	■ **About HeROS**: Open information about the operating system of the control ■ **About Xfce**: Open information on the Window manager
Tools	■ **Switch-off**: Shut-down or restart ■ **Screenshot**: Create screenshots ■ **File Manager**: Available only to authorized specialists ■ **Document Viewer**: Display and print files (e.g., PDF files) ■ **Geeqie**: Open, manage, and print graphics ■ **Gnumeric**: Open, edit, and print tables ■ **IDS Camera Manager**: Manage cameras connected to the control ■ **keypad horizontal**: Open virtual keyboard ■ **keypad vertical**: Open virtual keyboard ■ **Leafpad**: Open and edit text files ■ **NC Control**: Start or stop the NC software independently of the operating system ■ **NC/PLC Backup** **Further information:** "Backup and restore", Page 1955 ■ **NC/PLC Restore** **Further information:** "Backup and restore", Page 1955 ■ **QupZilla**: Alternative web browser for touch operation ■ **Real VNC viewer**: Define the settings for external software accessing the control for e.g. maintenance purposes ■ **Remote Desktop Manager** **Further information:** "Remote Desktop Manager window (option 133)", Page 1944 ■ **Ristretto**: Open graphics ■ **TNCguide**: Open help files in CHM format ■ **TouchKeyboard**: Open keyboard for touch operation ■ **Web Browser**: Start the web browser ■ **Xarchiver**: Extract or compress directories
Searching	Full-text search of individual functions

Taskbar



CAD-Viewer opened in the third desktop with taskbar shown and active HEROS menu

The taskbar consists of the following areas:

1 Workspaces
2 HEROS menu
  Further information: "Description of function", Page 1964
3 Opened applications, e.g.:
  - Control interface
  - **CAD-Viewer**
  - Window of HEROS functions

  You can move the opened applications into any other workspaces.
4 Widgets
  - Calendar
  - Status of the firewall
    Further information: "Firewall", Page 1950
  - Network status
    Further information: "Ethernet interface", Page 1925
  - Notifications
  - Shut down or restart the operating system

Window Manager

With the Window Manager you manage functions of the HEROS operating system as well as windows opened in the third desktop, such as the **CAD-Viewer**.

The control features the Xfce window manager. Xfce is a standard application for UNIX-based operating systems, and is used to manage graphical user interfaces. The following functions are possible with the window manager:

- Display a taskbar for switching between various applications (user interfaces)
- Manage an additional desktop, on which special applications from your machine manufacturer can run
- Control the focus between NC software applications and those of the machine manufacturer
- You can change the size and position of pop-up windows. It is also possible to close, minimize and restore pop-up windows

If a window is opened in the third desktop, the control displays the **Window Manager** icon in the information bar. You can switch between the open applications by selecting the icon.

You can minimize the control's user interface by pulling down from the information bar. The TNC bar and the OEM bar remain visible.

Further information: "Areas of the control's user interface", Page 89

Notes

- If a window is opened in the third desktop, the control displays an icon in the information bar.

  Further information: "Areas of the control's user interface", Page 89
- The machine manufacturer determines the scope of function and behavior of the window manager.
- The control shows a star in the upper left of the screen if an application of the window manager or the window manager itself has caused an error. In this case, switch to the window manager and correct the problem. If required, refer to your machine manual.

39.3 Serial data transfer

Application

The TNC7 automatically uses the LSV2 transmission protocol for serial data transfer. All parameters of the LSV2 protocol are invariably fixed except for the baud rate in the machine parameter **baudRateLsv2** (no. 106606).

Description of function

The machine parameter **RS232** (no. 106700) allows you to define another transmission type (interface). The settings described below are effective only for the respective newly defined interface.

Further information: "Machine parameters", Page 1958

In the machine parameters that then appear you can define the following settings:

Machine parameters	Setting
baudRate (no. 106701)	Data transfer rate (baud rate) Input: **BAUD_110**, **BAUD_150**, **BAUD_300 BAUD_600**, **BAUD_1200**, **BAUD_2400**, **BAUD_4800**, **BAUD_9600**, **BAUD_19200**, **BAUD_38400**, **BAUD_57600**, **BAUD_115200**
protocol (no. 106702)	Communications protocol ■ **STANDARD**: Standard data transmission, line-by-line ■ **BLOCKWISE**: Packet-based data transfer ■ **RAW_DATA**: Transmission without protocol (purely character-by-character) Input: **STANDARD**, **BLOCKWISE**, **RAW_DATA**
dataBits (no. 106703)	Data bits in each transferred character Input: **7 Bit**, **8 Bit**
parity (no. 106704)	Parity bit used to check for transmission errors ■ **NONE**: No parity, no error detection ■ **EVEN**: Even parity, error if the number of bits set is odd ■ **ODD**: Odd parity, error if the number of bits set is even Input: **NONE**, **EVEN**, **ODD**
stopBits (no. 106705)	The start bit and one or two stop bits enable the receiver to synchronize each transmitted character during serial data transmission. Input: **1 Stop-Bit**, **2 Stop-Bits**
flowControl (no. 106706)	By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking. ■ **NONE**: No data-flow check ■ **RTS_CTS**: Hardware handshaking, transmission stop is active through RTS ■ **XON_XOFF**: Software handshaking, transmission stop is active through DC3 Input: **NONE**, **RTS_CTS**, **XON_XOFF**
fileSystem (no. 106707)	File system for the serial interface ■ **EXT**: Minimum file system for printers or non-HEIDENHAIN transmission software ■ **FE1**: Communication with TNCserver or an external floppy disk unit If you require no special file system, this machine parameter is not needed. Input: **EXT**, **FE1**
bccAvoidCtrlChar (no. 106708)	The BCC is a block check character. The BCC is optionally added to a transfer block to simplify error detection. ■ **TRUE**: The BCC does not correspond to any control character ■ **FALSE**: Function not active Input: **TRUE**, **FALSE**

Machine parameters	Setting
rtsLow (no. 106709)	This optional parameter determines the level of the RTS line in the idle state. ■ **TRUE**: Level is **LOW** in idle state ■ **FALSE**: Level is **HIGH** in idle state Input: **TRUE**, **FALSE**
noEotAfterEtx (no. 106710)	This optional parameter sets whether an EOT character (End of Transmission) is to be transmitted after receiving an ETX character (End of Text). ■ **TRUE**: The EOT character is not sent ■ **FALSE**: The EOT character is sent Input: **TRUE**, **FALSE**

Example

In order to use the TNCserver PC software for data transfer, define the following settings in the machine parameter **RS232** (no. 106700):

Parameters	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transmission protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Type of handshake	RTS_CTS
File system for file operations	FE1

TNCserver is part of the TNCremo software for PCs.
"PC software for data transfer"

39.4 PC software for data transfer

Application

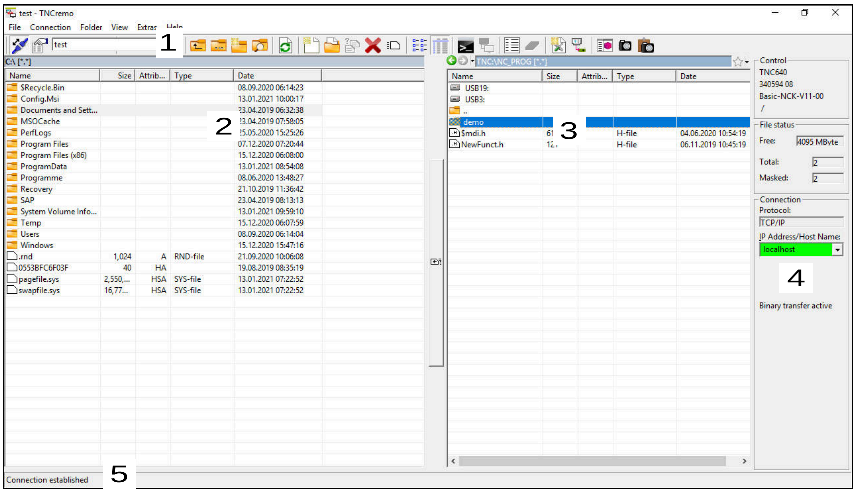
HEIDENHAIN offers the TNCremo software for connecting a Windows PC to a HEIDENHAIN control in order to transfer data.

Requirements

- PC operating system:
  - Windows 7
  - Windows 8
  - Windows 10
- PC RAM: 2 GB
- Free PC hard-disk space: 15 MB
- An available serial interface or connection to the control's network

Description of function

The TNCremo data transfer software provides the following areas:



1 Toolbar

This area provides the most important TNCremo functions.

2 File list of PC

In this area, TNCremo displays all of the folders and files of the connected drive (e.g., hard disk of a Windows PC or a USB flash drive).

3 File list of control

In this area, TNCremo displays all of the folders and files of the connected drive of the control.

4 Status display

In the status display, TNCremo shows information about the current connection.

5 Connection status

The connection status indicates whether a connection is currently active.

> For more information, refer to the integrated help system of TNCremo.
>
> You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

Notes

- When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user identification.
- You can download the current version of the TNCremo software from the **HEIDENHAIN website**.

39.5 Data backup

Application

If you create or modify files on the control, then you should back up these files periodically.

Related topics

- File management
  Further information: "File management", Page 1066

Description of function

With the functions **NC/PLC Backup** and **NC/PLC Restore** you can create back-up files for specific directories or even an entire drive, and restore them as needed. You should store these backup files on an external storage medium.

Further information: "Backup and restore", Page 1955

You have the following options for transferring files from the control:

- TNCremo
  With TNCremo you can transfer files from the control to a computer.
  Further information: "PC software for data transfer", Page 1970
- External drive
  You can transfer files from the control directly to an external drive.
  Further information: "Network drives on the control", Page 1922
- External data carriers
  You can store the files on external data carriers or use external data carriers to transfer the files.
  Further information: "USB devices", Page 1077

Notes

- You should back-up all machine-specific data, such as the PLC program or machine parameters. Consult your machine manufacturer about this.
- You must transmit files with the extensions PDF, XLS, ZIP, BMP, GIF, JPG and PNG in binary format from the PC to the control's hard disk.
- Backing up all files of the internal memory can take several hours. If required, perform the backup during a time when you don't need the machine.
- Periodically delete files that are no longer required. This ensures that the control has enough memory available for system files, such as the tool table.
- HEIDENHAIN recommends having the hard disk inspected after three to five years. After this time, and depending on the operating conditions (e.g., vibration loads), you must expect increased failure rates.

39.6 Opening files with additional software

Application

The control provides several additional software programs for opening and editing standard file types:

Related topics

- File types
  Further information: "File types", Page 1071

Description of function

The control offers tools for the following file types:

File type	Tool
PDF	Document Viewer
XSLX (XSL) CSV	Gnumeric
INI A TXT	Leafpad
HTM/HTML	Web browser For networks and the Internet, the machine manufacturer or network administrator must guarantee that the control is protected against viruses and malware (e.g., by a firewall).
ZIP	Xarchiver
BMP GIF JPG/JPEG PNG	Ristretto or Geeqie Ristretto can only open graphics files. Geeqie can also edit and print graphics.
OGG	Parole With Parole you can open the file types OGA, OGG, OGV and OGX. The Fuendo Codec Pack (available for payment) is needed only for other formats, such as MP4 files.

If you double-tap or double-click a file in the file manager, the control automatically starts the file with the correct tool. If more than one tool is possible for a file, the control displays a selection window.

The control opens the tools in the third desktop.

39.6.1 Opening tools

To open a tool:

- ▶ Select the HEIDENHAIN icon in the taskbar
- > The control opens the HEROS menu.
- ▶ Select **Tools**
- ▶ Select the tool, e.g. Leafpad
- > The control opens the tool in its own workspace.

Notes

- You can also open several tools from the **Desktop menu** workspace.
- Use the **ALT+TAB** key combination to switch between open workspaces.
- More information on how to use the various tools is provided within the respective tool under Help.
- After starting, the **web browser** checks at regular intervals whether updates are available.

  If you want to update the **web browser**, then you must deactivate the SELinux security software during this time and establish a connection to the Internet. Reactivate SELinux after the update!

  Further information: "SELinux security software", Page 1921

39.7 Network configuration with Advanced Network Configuration

Application

Using the **Advanced Network Configuration**, you can add, edit or remove profiles for the network connection.

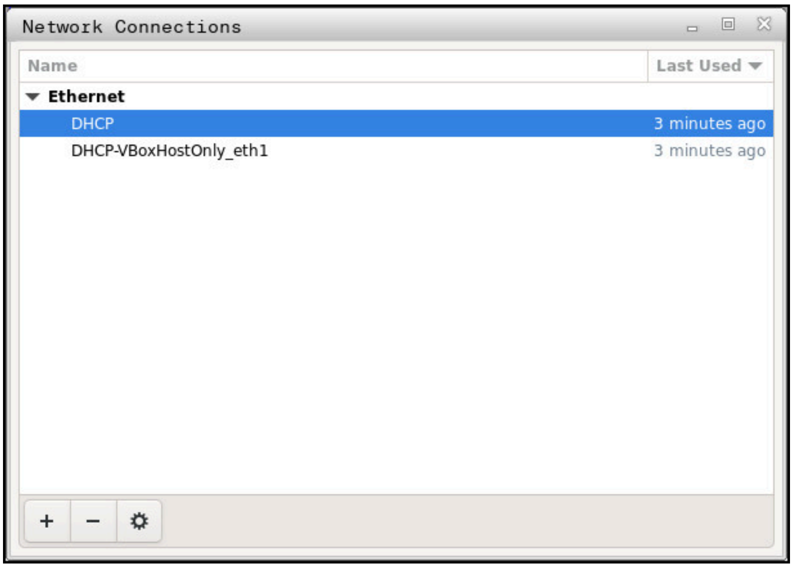
Related topics

- Network settings

  Further information: "Network settings window", Page 1927

Description of function

When you select the **Advanced Network Configuration** application in the HEROS menu, the control opens the **Network connections** window.



Network connections window

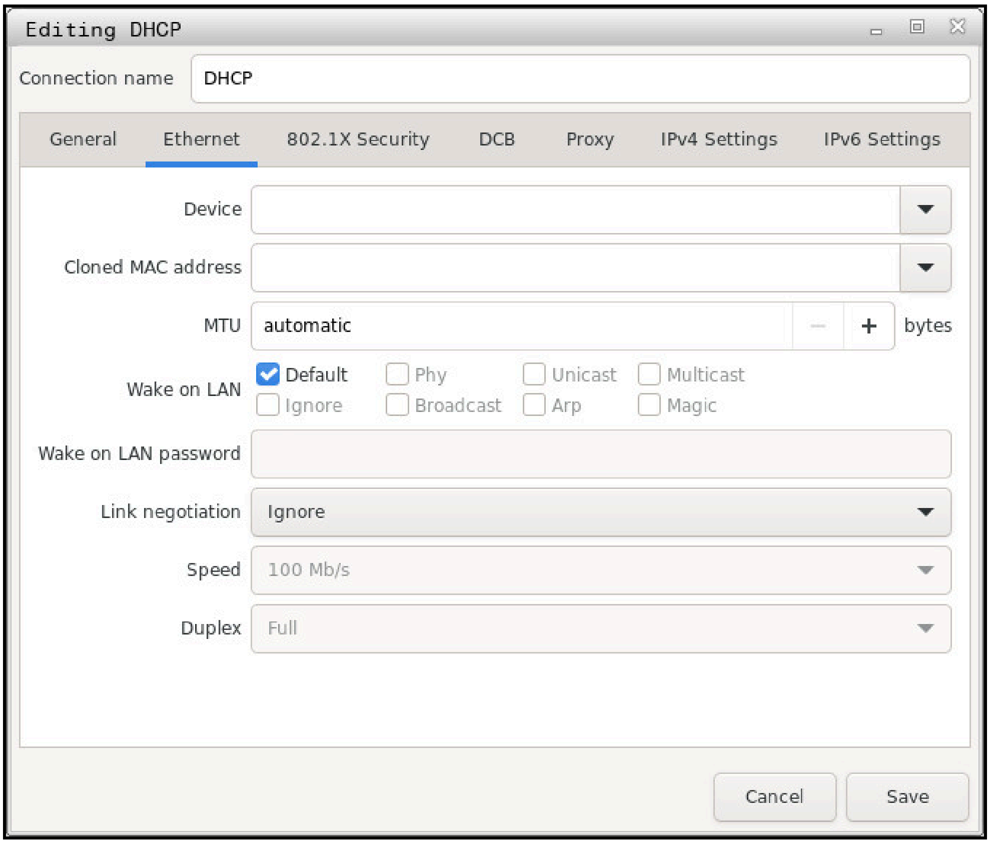
Symbols in the Network connections window

The following symbols are shown in the **Network connections** window:

Symbol	Function
+	Add network connection
−	Remove network connection
	Edit network connection The control opens the Editing network connection window. **Further information:** "Editing network connection window", Page 1975

39.7.1 Editing network connection window

In the **Editing network connection** window, the control shows the connection name of the network connection in the upper area. You can change the name.



Editing network connection window

General tab

The **General** tab contains the following settings:

Setting	Meaning
Connect automatically with priority	If you are using several profiles, you can define an order of priority for the connection here. The control connects the network with the highest priority first. Input: **-999...999**
All users may connect to this network	Here you can enable the selected network for all users.
Automatically connect to VPN	Currently no function
Metered connection	Currently no function

Ethernet tab

The **Ethernet** tab contains the following settings:

Setting	Meaning
Device	Here you can select the Ethernet interface. If you do not select an Ethernet interface, this profile can be used for any Ethernet interface. Selection by means of a selection window
Cloned MAC address	Currently no function
MTU	Here you can define the maximum package size in bytes. Input: **Automatic**, **1...10000**
Wake on LAN	Currently no function
Wake on LAN password	Currently no function
Link negotiation	Here you have to configure the settings for the Ethernet connection: ■ **Ignore** Retain the configurations already existing on the device. ■ **Automatic** The speed and duplex settings are configured automatically for the connection. ■ **Manual** Configure the speed and duplex settings for the connection manually. Selection by means of a selection window
Speed	Here you have to select the speed settings: ■ **10 Mb/s** ■ **100 Mb/s** ■ **1 Gb/s** ■ **10 Gb/s** Only with selection **Link negotiation Manual** Selection by means of a selection window
Duplex	Here you have to select the duplex setting: ■ **Half** ■ **Full** Only with selection **Link negotiation Manual** Selection by means of a selection window

802.1X Security tab

Currently no function

DCB tab

Currently no function

Proxy tab

Currently no function

IPv4 Settings tab

The **IPv4 Settings** tab contains the following settings:

Setting	Meaning
Method	Here you have to select a network connection method: ■ **Automatic (DHCP)** If the network uses a DHCP server for IP address assignment ■ **Automatic (DHCP) addresses only** If the network uses a DHCP server for IP address assignment, but you are assigning the DNS server manually ■ **Manual** Assign the IP address manually ■ **Link-Local Only** Currently no function ■ **Shared to other computers** Currently no function ■ **Disabled** Deactivate IPv4 for this connection
Additional static addresses	Here you can add static IP addresses that will be set up in addition to the IP addresses that are assigned automatically. Only with **Method Manual**
Additional DNS servers	Here you can add the IP addresses of DNS servers that are used to resolve computer names. Separate multiple IP addresses by commas. Only with **Methods Manual** and **Automatic (DHCP) addresses only**
Additional search domains	Here you can add domains used by computer names. Separate multiple domains by commas. Only with **Method Manual**
DHCP client ID	Currently no function
Require IPv4 addressing for this connection to complete	Currently no function

IPv6 Settings tab

Currently no function

40

Overviews

40.1 Pin layout and cables for data interfaces

40.1.1 V.24/RS-232-C interface for HEIDENHAIN devices

The interface complies with the requirements of EN 50178 for Secure separation from the power grid.

Control		25-pin: VB 274545-xx			9-pin: VB 366964-xx		
Male	**Assignment**	**Male**	**Color**	**Female**	**Female**	**Color**	**Female**
1	Do not assign	1	White/Brown	1	1	Red	1
2	RXD	3	Yellow	2	2	Yellow	3
3	TXD	2	Green	3	3	White	2
4	DTR	20	Brown	8	4	Brown	6
5	Signal GND	7	Red	7	5	Black	5
6	DSR	6		6	6	Violet	4
7	RTS	4	Gray	5	7	Gray	8
8	CTR	5	Pink	4	8	White/Green	7
9	Do not assign	8	Violet	20	9	Green	9
Housing	External shield	Housing	External shield	Housing	Housing	External shield	Housing

40.1.2 Ethernet interface RJ45 socket

Maximum cable length:

- 100 m unshielded
- 400 m shielded

Pin	Signal
1	TX+
2	TX–
3	RX+
4	Vacant
5	Vacant
6	RX–
7	Vacant
8	Vacant

40.2 Machine parameters

The following list shows the machine parameters that you can edit with the code number 123.

Related topics

- Editing machine parameters with the **MPs for setters** application

  Further information: "Machine parameters", Page 1958

40.2.1 List of user parameters

Refer to your machine manual.

- The machine manufacturer can make additional machine-specific parameters available as user parameters, so that you can configure the functions that are available.
- The machine manufacturer can adapt the structure and contents of the user parameters. The display on your machine may be different.

Depiction in the configuration editor	MP number	Page
DisplaySettings Settings for screen displays		-
CfgDisplayData Settings for screen displays	100800	1993
axisDisplay Sequence and rules for displayed axes	100810	1993
x Key name of the axis		-
axisKey Key name of the axis	100810. [Index].01501	1993
name Axis designation	100810. [Index].01502	1993
rule Display rule for the axes	100810. [Index].01503	1993
axisDisplayRef Sequence and rules for display axes before crossing the reference marks	100811	1994
x Key name of the axis		-
axisKey Key name of the axis	100811. [Index].01501	1994
name Axis designation	100811. [Index].01502	1994
rule Display rule for the axes	100811. [Index].01503	1994
positionWinDisplay Type of position display in the positioning window	100803	1995
statusWinDisplay Type of position display in the status display	100804	1995
decimalCharacter Definition of decimal separator for position display	100805	1996
axisFeedDisplay Display of the feed rate in Manual Operation and El. Handwheel modes of operation	100806	1996

Depiction in the configuration editor	MP number	Page
spindleDisplay Display of spindle position in the position display	100807	1996
hidePresetTable Disable the PRESET MANAGEMENT soft key	100808	1997
displayFont Font size for program display in the operating modes Program Run Full Sequence, Program Run Single Block, and Positioning with Manual Data Input.	100812	1997
iconPrioList Sequence of icons in the display	100813	1997
compatibilityBits Settings for display behavior	100815	1998
axesGridDisplay Axes as list or group in the position display.	100806	1998
CfgPosDisplayPace Display step for the individual axes	101000	1998
xx Display step for position display in [mm] or [°]		-
displayPace Display step for position display in [mm] or [°]	101001	1998
displayPaceInch Display step for position display in [inch]	101002	1999
CfgUnitOfMeasure Definition of unit of measure in effect for display	101100	1999
unitOfMeasure Unit of measure for display and user interface	101101	1999
CfgProgramMode Format of the NC programs and cycle display	101200	1999
programInputMode MDI: Program entry in HEIDENHAIN Klartext format or ISO format	101201	1999
CfgDisplayLanguage Definition of the NC and PLC conversational language	101300	2000
ncLanguage NC conversational language	101301	2000
applyCfgLanguage Load the language of the NC control	101305	2001
plcDialogLanguage PLC conversational language	101302	2001
plcErrorLanguage PLC error message language	101303	2001

Depiction in the configuration editor	MP number	Page
helpLanguage Language for online help	101304	2002
CfgStartupData Behavior during control startup	101500	2003
powerInterruptMsg Acknowledge the Power interrupted message	101501	2003
opMode Operating mode that is switched to when the control has fully booted	101503	2003
subOpMode Submode to be activated for the operating mode entered in 'opMode'	101504	2003
CfgClockView Display mode for time of day	120600	2004
displayMode Display mode for time of day on the screen	120601	2004
timeFormat Time format of digital clock	120602	2004
CfgInfoLine Link row on/off	120700	2004
infoLineEnabled Enable/disable info line	120701	2004
CfgGraphics Settings for 3-D simulation graphics	124200	2005
modelType Model type of the 3D simulation graphics	124201	2005
modelQuality Model quality of the 3D simulation graphics	124202	2005
clearPathAtBlk Reset tool paths for new BLK FORM	124203	2005
extendedDiagnosis Write graphics journal files after restart	124204	2006
CfgPositionDisplay Settings for the digital readout	124500	2006
progToolCallDL Position display with TOOL CALL DL	124501	2006
CfgTableEditor Table editor configuration	125300	2006
deleteLoadedTool Behavior when deleting tools from the pocket table	125301	2007
indexToolDelete Behavior when deleting a tool's index entries	125302	2007

Depiction in the configuration editor	MP number	Page
showResetColumnT Show the RESET T soft key	125303	2007
CfgDisplayCoordSys Setting the coordinate systems for the display	127500	2007
transDatumCoordSys Coordinate system for the datum shift	127501	2008
CfgGlobalSettings GPS display settings	128700	2008
enableOffset Show offset in the GPS dialog	128702	2008
enableBasicRot Show an additive basic rotation in the GPS dialog	128703	2008
enableShiftWCS Show shift of W-CS in the GPS dialog	128704	2008
enableMirror Show mirror image in the GPS dialog	128712	2009
enableShiftMWCS Show shift of mW-CS in the GPS dialog	128711	2009
enableRotation Show rotation in the GPS dialog	128707	2009
enableFeed Show feed rate the GPS dialog	128708	2009
enableHwMCS M-CS coordinate system is selectable	128709	2009
enableHwWCS W-CS coordinate system is selectable	128710	2010
enableHwMWCS mW-CS coordinate system is selectable	128711	2010
enableHwWPLCS WPL-CS coordinate system is selectable	128712	2010
CfgRemoteDesktop Settings for Remote Desktop connections	100800	2010
connections List of Remote Desktop connections to be displayed	133501	2011
title Name of the OEM operating mode	133502	2011
dialogRes Name of a text	133502.00501	2011
text Language-sensitive text	133502.00502	2011

Depiction in the configuration editor	MP number	Page
icon Path/name for optional icon graphic file	133503	2011
locations List with positions where this Remote Desktop connection is displayed	133504	2011
x Operating mode		-
opMode Operating mode	133504.[Index].133401	2012
subOpMode Optional submode for the operating mode specified in 'opMode'	133504.[Index].133402	2012
ProbeSettings Configuration of the tool calibration		-
CfgTT Configuration of the tool calibration	122700	2013
TT140_x M function for spindle orientation		-
spindleOrientMode M function for spindle orientation	122704	2013
probingRoutine Probing routine	122705	2013
probingDirRadial Probing direction for tool radius measurement	122706	2013
offsetToolAxis Distance from lower edge of tool to upper edge of stylus	122707	2014
rapidFeed Rapid traverse in probing cycle for TT tool touch probe	122708	2014
probingFeed Probing feed rate for tool measurement with non-rotating tool	122709	2014
probingFeedCalc Calculation of the probing feed rate	122710	2014
spindleSpeedCalc Speed determination method	122711	2014
maxPeriphSpeedMeas Maximum permissible surface speed of the tool edge for radius measurement	122712	2015
maxSpeed Maximum permissible speed during tool measurement	122714	2015

Depiction in the configuration editor	MP number	Page
measureTolerance1 Maximum permissible measuring error for tool measurement with rotating tool (first measurement error)	122715	2015
measureTolerance2 Maximum permissible measuring error for tool measurement with rotating tool (second measurement error)	122716	2015
stopOnCheck NC stop during tool check	122717	2015
stopOnMeasurement NC stop during tool measurement	122718	2016
adaptToolTable Change the tool table during tool check and tool measurement	122719	2016
CfgTTRoundStylus Configuration of a round stylus	114200	2016
TT140_x Coordinates of the TT tool touch probe stylus contact center with respect to the machine datum		-
centerPos Coordinates of the TT tool touch probe stylus contact center with respect to the machine datum	114201	2016
safetyDistToolAx Safety clearance around the probe contact of the TT tool touch probe for pre-positioning in the tool-axis direction	114203	2017
safetyDistStylus Safety zone around the stylus for pre-positioning	114204	2017
CfgTTRectStylus Configuration of a rectangular stylus	114300	2017
TT140_x Coordinates of the stylus center		-
centerPos Coordinates of the stylus center	114313	2017
safetyDistToolAx Set-up clearance above the stylus for pre-positioning	114317	2017
safetyDistStylus Safety zone around the stylus for pre-positioning	114318	2017
ChannelSettings Active kinematics		-

Depiction in the configuration editor	MP number	Page
CH_xx Active kinematics		-
CfgActivateKinem Active kinematics	204000	2019
kinemToActivate Kinematics to be activated / Active kinematics	204001	2019
kinemAtStartup The kinematics to be activated during control start-up	204002	2019
CfgNcPgmBehaviour Specify the behavior of the NC program.	200800	2019
operatingTimeReset Reset the machining time when program starts.	200801	2019
plcSignalCycle PLC signal for the number of the pending machining cycle	200803	2020
CfgGeoTolerance Geometry tolerances	200900	2020
circleDeviation Permissible deviation of the radius	200901	2020
threadTolerance Permissible deviation in successive threads	200902	2020
moveBack Reserve for retraction movements	200903	2020
CfgGeoCycle Configuration of the fixed cycles	201000	2021
pocketOverlap Overlap factor for pocket milling	201001	2021
posAfterContPocket Traverse after machining the contour pocket	201007	2021
displaySpindleErr Display the Spindle is not rotating error message if M3/M4 is not active	201002	2021
displayDepthErr Display the Check the depth sign error message	201003	2021
apprDepCylWall Behavior when moving to wall of slot in the cylinder surface	201004	2022

Depiction in the configuration editor	MP number	Page
mStrobeOrient M function for spindle orientation in machining cycles	201005	2022
suppressPlungeErr Do not show 'Plunging type is not possible' error message	201006	2022
restoreCoolant Behavior of M7 and M8 with Cycles 202 and 204	201008	2022
facMinFeedTurnSMAX Automatic feed rate reduction after attaining SMAX	201009	2023
suppressResMatlWar Do not show "Residual material" warning	201010	2023
CfgStretchFilter Geometry filter for filtering out linear elements	201100	2023
filterType Type of stretch filter	201101	2024
tolerance Maximum distance of the filtered to the unfiltered contour	201102	2024
maxLength Maximum length of the distance resulting from filtering	201103	2024
CfgThreadSpindle	113600	2024
sourceOverride Effective override potentiometer for feed rate during thread cutting	113603	2025
thrdWaitingTime Waiting time at reversal point in thread base	113601	2025
thrdPreSwitchTime Advanced switching time of spindle	113602	2025
limitSpindleSpeed Limit of spindle speed with Cycles 17, 207 and 18	113604	2025
CfgEditorSettings Settings for the NC editor		-
CfgEditorSettings Settings for the NC editor	105400	2027
createBackup Generate a backup file *.bak	105401	2027
deleteBack Behavior of the cursor after deletion of lines	105402	2027

Depiction in the configuration editor	MP number	Page
cursorAround Behavior of the cursor on the first or last line	105403	2027
lineBreak Line break on NC blocks with more than one line	105404	2027
stdTNChelp Activate help graphics when entering cycle data	105405	2028
toggleCyclDef Behavior of the cycle soft key row after a cycle entry	105406	2028
warningAtDEL Confirmation request when deleting an NC block.	105407	2028
maxLineGeoSearch Line number up to which a test of the NC program is to be run.	105408	2028
blockIncrement ISO programming: Block number increment	105409	2029
useProgAxes Specify programmable axes	105410	2029
enableStraightCut Allow or lock paraxial positioning blocks	105411	2029
maxLineCommandSrch Number of lines for searching for identical syntax elements	105412	2029
noParaxMode Allow/lock FUNCTION PARAXCOMP/PARAX-MODE via soft key	105413	2029
CfgPgmMgt Settings for the file management		-
CfgPgmMgt Settings for the file management	122100	2030
dependentFiles Display of dependent files	122101	2030
CfgProgramCheck Settings for tool-usage files		-
CfgProgramCheck Settings for tool-usage files	129800	2031
autoCheckTimeOut Timeout for creation of tool-usage files	129803	2031
autoCheckPrg Create tool-usage file for NC program	129801	2031
autoCheckPal Create pallet-usage files	129802	2031

Depiction in the configuration editor	MP number	Page
CfgUserPath Paths for the end user		-
CfgUserPath Paths for the end user	102200	2032
ncDir List of drives and/or directories	102201	2032
fn16DefaultPath Default output path for the FN16: F-PRINT function in the Program Run operating modes	102202	2032
fn16DefaultPathSim Default output path for the FN16: F-PRINT function in the Programming and Test Run operating modes	102203	2032
serialInterfaceRS232 Data record belonging to the serial port		-
CfgSerialPorts Data record belonging to the serial port	106600	2033
activeRs232 Enable the RS-232 interface in the program manager	106601	2033
baudRateLsv2 Data transfer rate for LSV2 communication in baud	106606	2033
CfgSerialInterface Definition of data records for the serial ports	106700	2033
RS232 Data transfer rate for communication in baud		-
baudRate Data transfer rate for communication in baud	106701	2034
protocol Communications protocol	106702	2034
dataBits Data bits in each transferred character	106703	2034
parity Type of parity checking	106704	2035
stopBits Number of stop bits	106705	2035
flowControl Type of data-flow checking	106706	2035
fileSystem File system for file operation via serial interface	106707	2035

Depiction in the configuration editor	MP number	Page
bccAvoidCtrlChar Avoid control characters in the block check character (BCC)	106708	2036
rtsLow Idle state of the RTS line	106709	2036
noEotAfterEtx Behavior after reception of an ETX control character	106710	2036
Monitoring Monitoring settings for the user		-
CfgMonUser Monitoring settings for the user	129400	2037
enforceReaction The configured error reactions are enforced	129401	2037
showWarning Display warnings of monitoring tasks	129402	2037
CfgMonMbSection CfgMonMbSection defines monitoring tasks for a certain section of an NC program	02400	2037
tasks List of monitoring tasks to be performed	133701	2037
CfgMachineInfo General information of the machine operator		-
CfgMachineInfo General information of the machine operator	131700	2038
machineNickname Custom name (nickname) of the machine	131701	2038
inventoryNumber Inventory number or ID	131702	2038
image Photo or image of the machine	131703	2038
location Machine location	131704	2038
department Department or division	131705	2038
responsibility Responsible for the machine	131706	2038
contactEmail Contact email address	131707	2039
contactPhoneNumber Contact phone number	131708	2039

40.2.2 Details about the user parameters

Explanations about the detailed view of user parameters:

- The indicated path corresponds to the machine parameter structure that you see after entering the machine manufacturer code number. With this information you can also find the desired machine parameter in the alternative structure. With the machine parameter numbers you can search for the machine parameters independently of the structure.
- The entry after iTNC shows the machine parameter number on the iTNC 530.

DisplaySettings

CfgDisplayData	100800
Settings for screen displays	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData
Structural element:	

axisDisplay	100810
Sequence and rules for displayed axes	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplay
Input:	Up to 24 lists with parameters **MP_axisKey**, **MP_name** and **MP_rule**. Specifies sequence and rules for the display of an axis.

axisKey	100810. [Index].01501
Key name of the axis	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplay ▶ [Index] ▶ axisKey
Input:	Select the key name of the axis for which this display setting is valid. The key names of the axes are taken from the configuration entity CfgAxis and displayed in a selection menu.

name	100810. [Index].01502
Axis designation	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplay ▶ [Index] ▶ name
Input:	max. 2 Characters Defines the axis name, also used for the display alternatively to the key name from CfgAxis. If the parameter is not set, the TNC7 displays the key name.

rule	100810. [Index].01503
Display rule for the axes	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplay ▶ [Index] ▶ rule
Input:	Specifies the condition under which the axis is displayed: **ShowAlways** Axis is always shown. The display location remains reserved even if no values for the axis can be displayed, for example if the axis is not contained in the current kinematic model.

IfKinem

The axis is only shown if used as axis or spindle in the active kinematic model.

IfKinemAxis

Axis only shown if used as axis in the active kinematics model.

IfNotKinemAxis

The axis is only shown if it is not used as an axis in the active kinematics model (e.g. as spindle).

Never

The axis is not shown.

axisDisplayRef		100811
Sequence and rules for display axes before crossing the reference marks		
	Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplayRef
	Input:	Up to 24 lists with parameters **MP_axisKey**, **MP_name** and **MP_rule**. Specifies the sequence and the rules for the display of axes if the position display is set to REF values (also applies when traversing to the reference point). If this list is empty, the entries from axisDisplay will be used. The top-most entry corresponds to the top-most position.

axisKey		100811.[Index].01501
Key name of the axis		
	Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplayRef ► [Index] ► axisKey
	Input:	Select the key name of the axis for which this display setting is valid. The key names of the axes are taken from the configuration entity CfgAxis and displayed in a selection menu.

name		100811.[Index].01502
Axis designation		
	Path:	System ► DisplaySettings ► CfgDisplayData ► axisDisplayRef ► [Index] ► name
	Input:	max. 2 Characters Defines the axis name, also used for the display alternatively to the key name from CfgAxis. If the parameter is not set, the TNC7 displays the key name.

rule		100811.[Index].01503
Display rule for the axes		

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisDisplayRef ▶ [Index] ▶ rule
Input:	Specifies the condition for displaying the axis: **ShowAlways** Axis is always shown. **IfKinem** Axis only shown if used as axis or spindle in the active kinematics model. **IfKinemAxis** Axis only shown if used as axis in the active kinematics model. **IfNotKinemAxis** The axis is only shown if it is not used as an axis in the active kinematics model (e.g. as spindle). **Never** The axis is not shown.

positionWinDisplay 100803

Type of position display in the positioning window

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ positionWinDisplay
Input:	**NOML** Nominal position **ACTL** Actual position **REF ACTL** Actual position referenced to the machine datum **REF NOML** Nominal position referenced to the machine datum **LAG** Following error (servo lag) **ACTDST** Distance-to-go in the input system **REFDST** Distance-to-go in the machine system **M118** Traverse paths that were carried out with handwheel superimpositioning (M118)

statusWinDisplay 100804

Type of position display in the status display

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ statusWinDisplay
Input:	**NOML** Nominal position **ACTL**

Actual position

REF ACTL

Actual position referenced to the machine datum

REF NOML

Nominal position referenced to the machine datum

LAG

Following error (servo lag)

ACTDST

Distance-to-go in the input system

REFDST

Distance-to-go in the machine system

M118

Traverse paths that were carried out with handwheel superimpositioning (M118)

decimalCharacter 100805

Definition of decimal separator for position display

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ decimalCharacter
Input:	"." ","
iTNC 530:	7280

axisFeedDisplay 100806

Display of the feed rate in **Manual Operation** and **El. Handwheel** modes of operation

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axisFeedDisplay
Input:	**at axis key** Display of the feed rate only if an axis direction key is pressed. The axis-specific feed rate from CfgFeedLimits/**MP_manualFeed** is displayed. **always minimum** Display of the feed rate also before an axis direction key is pressed (lowest value from CfgFeedLimits/**MP_manualFeed**) for all axes.
iTNC 530:	7270

spindleDisplay 100807

Display of spindle position in the position display

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ spindleDisplay
Input:	**during closed loop** Display of spindle position only if the spindle is servo controlled **during closed loop and M5**

Display of spindle position if the spindle is servo controlled and M5 is active

during closed loop or M5 or tapping

Display of spindle position if the spindle is servo controlled or M5 is active or when tapping

hidePresetTable	100808

Disable the **PRESET MANAGEMENT** soft key

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ hidePresetTable
Input:	**TRUE** Access to the preset table is locked; the soft key is dimmed **FALSE** The preset table can be accessed via soft key

displayFont	100812

Font size for program display in the operating modes Program Run Full Sequence, Program Run Single Block, and Positioning with Manual Data Input.

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ displayFont
Input:	**FONT_APPLICATION_SMALL** Small font size. Same font size as in the Programming and Test Run operating modes. **FONT_APPLICATION_MEDIUM** Big font size.

iconPrioList	100813

Sequence of icons in the display

Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ iconPrioList
Input:	**BASIC_ROT** **ROT_3D** **TCPM** **ACC** **TURNING** **AFC** **S_PULSE** **MIRROR** **GPS** **RADCORR** **PARAXCOMP**

MON_FS_OVR

compatibilityBits	100815
Settings for display behavior	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ compatibilityBits
Input:	Bit ■ 0: in the small PLC window with half the width and without a bar graph, the characters are always shown in the small font size. ■ 1: in the small PLC window with half the width and with a bar graph, the characters are always shown in the large font size.

axesGridDisplay	100816
Axes as list or group in the position display.	
Path:	System ▶ DisplaySettings ▶ CfgDisplayData ▶ axesGridDisplay
Input:	The parameter specifies whether the axes in the position display are shown as a list or as a two-column grid. Possible settings: 0 to **0** Axis display as list (default) **Quantity (n)** Axis display as two-column grid with groups of n x 2 axes
iTNC 530:	7270

CfgPosDisplayPace	101000
Display step for the individual axes	
Path:	System ▶ DisplaySettings ▶ CfgPosDisplayPace
Structural element:	

displayPace	101001
Display step for position display in [mm] or [°]	
Path:	System ▶ DisplaySettings ▶ CfgPosDisplayPace ▶ [Key name of the axis] ▶ displayPace
Input:	**0.1** **0.05** **0.01** **0.005** **0.001** **0.0005** **0.0001**

	0.00005
	0.00001
	0.000005
	0.000001
iTNC 530:	7290.0-8

displayPaceInch 101002

Display step for position display in [inch]

Path:	System ▶ DisplaySettings ▶ CfgPosDisplayPace ▶ [Key name of the axis] ▶ displayPaceInch
Input:	**0.005** **0.001** **0.0005** **0.0001** **0.00005** **0.00001** **0.000005** **0.000001**
iTNC 530:	7290.0-8

CfgUnitOfMeasure 101100

Definition of unit of measure in effect for display

Path:	System ▶ DisplaySettings ▶ CfgUnitOfMeasure
Structural element:	

unitOfMeasure 101101

Unit of measure for display and user interface

Path:	System ▶ DisplaySettings ▶ CfgUnitOfMeasure ▶ unitOfMeasure
Input:	**metric** Metric measurement system **inch** Inches

CfgProgramMode 101200

Format of the NC programs and cycle display

Path:	System ▶ DisplaySettings ▶ CfgProgramMode
Structural element:	

programInputMode 101201

MDI: Program entry in HEIDENHAIN Klartext format or ISO format

Path:	System ▶ DisplaySettings ▶ CfgProgramMode ▶ programInputMode
Input:	**HEIDENHAIN** Program entry with HEIDENHAIN Klartext **ISO** Program entry according to ISO

CfgDisplayLanguage 101300

Definition of the NC and PLC conversational language

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage
Structural element:	

ncLanguage 101301

NC conversational language

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ ncLanguage
Input:	**ENGLISH** **GERMAN** **CZECH** **FRENCH** **ITALIAN** **SPANISH** **PORTUGUESE** **SWEDISH** **DANISH** **FINNISH** **DUTCH** **POLISH** **HUNGARIAN** **RUSSIAN** **CHINESE** **CHINESE_TRAD** **SLOVENIAN** **KOREAN** **NORWEGIAN** **ROMANIAN** **SLOVAK** **TURKISH**

iTNC 530:	7230.0

applyCfgLanguage 101305

Load the language of the NC control

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ applyCfgLanguage
Input:	When booting, the control checks whether the language settings of the operating system and the NC are the same. If the settings differ, the NC applies the language setting of the operating system. If the language defined in the machine parameters of the NC is to be used, then you must the set the parameter applyCfgLanguage to TRUE.

plcDialogLanguage 101302

PLC conversational language

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ plcDialogLanguage
Input:	**ENGLISH** **GERMAN** **CZECH** **FRENCH** **ITALIAN** **SPANISH** **PORTUGUESE** **SWEDISH** **DANISH** **FINNISH** **DUTCH** **POLISH** **HUNGARIAN** **RUSSIAN** **CHINESE** **CHINESE_TRAD** **SLOVENIAN** **KOREAN** **NORWEGIAN** **ROMANIAN** **SLOVAK** **TURKISH**
iTNC 530:	7230.1

plcErrorLanguage 101303

PLC error message language

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ plcErrorLanguage
Input:	**ENGLISH** **GERMAN** **CZECH** **FRENCH** **ITALIAN** **SPANISH** **PORTUGUESE** **SWEDISH** **DANISH** **FINNISH** **DUTCH** **POLISH** **HUNGARIAN** **RUSSIAN** **CHINESE** **CHINESE_TRAD** **SLOVENIAN** **KOREAN** **NORWEGIAN** **ROMANIAN** **SLOVAK** **TURKISH**
iTNC 530:	7230.2

helpLanguage 101304

Language for online help

Path:	System ▶ DisplaySettings ▶ CfgDisplayLanguage ▶ helpLanguage
Input:	**ENGLISH** **GERMAN** **CZECH** **FRENCH** **ITALIAN** **SPANISH** **PORTUGUESE** **SWEDISH** **DANISH**

	FINNISH
	DUTCH
	POLISH
	HUNGARIAN
	RUSSIAN
	CHINESE
	CHINESE_TRAD
	SLOVENIAN
	KOREAN
	NORWEGIAN
	ROMANIAN
	SLOVAK
	TURKISH
iTNC 530:	7230.3

CfgStartupData 101500

Behavior during control startup

Path:	System ▶ DisplaySettings ▶ CfgStartupData
Structural element:	

powerInterruptMsg 101501

Acknowledge the **Power interrupted** message

Path:	System ▶ DisplaySettings ▶ CfgStartupData ▶ powerInterruptMsg
Input:	**TRUE** Start-up is only continued after the message has been acknowledged. **FALSE** The **Power interrupted** message does not appear

opMode 101503

Operating mode that is switched to when the control has fully booted

Path:	System ▶ DisplaySettings ▶ CfgStartupData ▶ opMode
Input:	Enter here the GUI designator of the desired operating mode. See the Technical Manual for an overview of the permissible GUI designators. max. 500 Characters

subOpMode 101504

Submode to be activated for the operating mode entered in 'opMode'

Path:	System ▶ DisplaySettings ▶ CfgStartupData ▶ subOpMode

Input:	Enter here the GUI designator of the desired operating submode. See the Technical Manual for an overview of the permissible GUI designators. max. 500 Characters

CfgClockView 120600

Display mode for time of day

Path:	System ▶ DisplaySettings ▶ CfgClockView
Structural element:	

displayMode 120601

Display mode for time of day on the screen

Path:	System ▶ DisplaySettings ▶ CfgClockView ▶ displayMode
Input:	**Analog** Analog clock **Digital** Digital clock **Logo** OEM logo **Analog and logo** Analog clock and OEM logo **Digital and logo** Digital clock and OEM logo **Analog on logo** Analog clock that superimposes the OEM logo **Digital on logo** Digital clock that superimposes the OEM logo

timeFormat 120602

Time format of digital clock

Path:	System ▶ DisplaySettings ▶ CfgClockView ▶ timeFormat
Input:	Possible settings: **12 h format** Time in 12 hours format **24 h format** Time in 24 hours format

CfgInfoLine 120700

Link row on/off

Path:	System ▶ DisplaySettings ▶ CfgInfoLine
Structural element:	

infoLineEnabled 120701

Enable/disable info line

Path:	System ▶ DisplaySettings ▶ CfgInfoLine ▶ infoLineEnabled
Input:	**OFF** The info line is disabled **ON** The info line below the operating mode display is enabled

CfgGraphics 124200

Settings for 3-D simulation graphics

Path:	System ▶ DisplaySettings ▶ CfgGraphics
Structural element:	

modelType 124201

Model type of the 3D simulation graphics

Path:	System ▶ DisplaySettings ▶ CfgGraphics ▶ modelType
Input:	**No Model** The model depiction is deactivated. Only the 3D line graphics are shown (lowest processor load, e.g. for fast testing of the NC program and ascertainment of program run times) **3D** Model depiction for complex operations (highest processor load, e.g. for turning or undercuts) **2.5D** Model depiction for 3-axis operations (medium processor load)

modelQuality 124202

Model quality of the 3D simulation graphics

Path:	System ▶ DisplaySettings ▶ CfgGraphics ▶ modelQuality
Input:	**very high** Very high model quality, the production result can be precisely judged. This setting requires the highest computing power. Block numbers and block end points can only be displayed in the 3D line graphics with this setting. **high** High model quality **medium** Medium model quality **low** Low model quality

clearPathAtBlk 124203

Reset tool paths for new BLK FORM

Path:	System ▶ DisplaySettings ▶ CfgGraphics ▶ clearPathAtBlk
Input:	**ON** With a new BLK FORM in the Test Run graphic, the tool paths are reset **OFF** With a new BLK FORM in the Test Run graphic, the tool paths are not reset

extendedDiagnosis 124204

Write graphics journal files after restart

Path:	System ▶ DisplaySettings ▶ CfgGraphics ▶ modelType
Input:	Activate diagnostic information for HEIDENHAIN (journal files) for the analysis of graphics problems. **OFF** Do not create journal files (default). **ON** Create journal files.

CfgPositionDisplay 124500

Settings for the digital readout

Path:	System ▶ DisplaySettings ▶ CfgPositionDisplay
Structural element:	

progToolCallDL 124501

Position display with TOOL CALL DL

Path:	System ▶ DisplaySettings ▶ CfgPositionDisplay ▶ progToolCallDL
Input:	**As Tool Length** Default behavior of the TNC7. The oversize DL programmed in the TOOL CALL block is taken into account as part of the tool length in the nominal position display. **As Workpiece Oversize** iTNC 530-compatible setting. The oversize DL programmed in the TOOL CALL block is not considered in the nominal position display. Thus, it is effective as workpiece oversize.

CfgTableEditor 125300

Table editor configuration

Path:	System ▶ TableSettings ▶ CfgTableEditor

Structural element:	Specifies properties and settings for the table editor.

deleteLoadedTool 125301

Behavior when deleting tools from the pocket table

Path:	System ▶ TableSettings ▶ CfgTableEditor ▶ deleteLoadedTool
Input:	Possible settings: **DISABLED** Tool deletion is not possible **WITH_WARNING** Tool deletion is possible; Note must be confirmed **WITHOUT_WARNING** Tool deletion is possible without confirmation
iTNC 530:	7263 Bit4, 7263 Bit5

indexToolDelete 125302

Behavior when deleting a tool's index entries

Path:	System ▶ TableSettings ▶ CfgTableEditor ▶ indexToolDelete
Input:	Possible settings: **ALWAYS_ALLOWED** Deletion of index entries is always possible **TOOL_RULES** Behavior depends on the setting of the parameter deleteLoadedTool
iTNC 530:	7263 Bit6

showResetColumnT 125303

Show the **RESET T** soft key

Path:	System ▶ TableSettings ▶ CfgTableEditor ▶ showResetColumnT
Input:	The parameter specifies whether the **RESET T** soft key will be offered when a pocket table is open in the table editor. **TRUE** The soft key is displayed. All tools can be deleted from the tool memory by the user. **FALSE** The soft key is not displayed.
iTNC 530:	7263 Bit3

CfgDisplayCoordSys 127500

Setting the coordinate systems for the display

Path:	System ▶ DisplaySettings ▶ CfgDisplayCoordSys

Structural element:	

transDatumCoordSys 127501

Coordinate system for the datum shift

Path:	System ▶ DisplaySettings ▶ CfgDisplayCoordSys ▶ transDatumCoordSys
Input:	The parameter specifies the coordinate system in which the datum shift is displayed. **WorkplaneSystem** Datum is displayed in the system of the tilted plane (WPL-CS) **WorkpieceSystem** Datum is displayed in the workpiece coordinate system (W-CS)

CfgGlobalSettings 128700

GPS display settings

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings
Structural element:	

enableOffset 128702

Show offset in the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableOffset
Input:	**OFF** Offset is not displayed **ON** Offset is displayed

enableBasicRot 128703

Show an additive basic rotation in the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableBasicRot
Input:	**OFF** Additive basic rotation is not displayed **ON** Additive basic rotation is displayed

enableShiftWCS 128704

Show shift of W-CS in the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableShiftWCS

Input:	**OFF** Shift of W-CS (workpiece coordinate system) is not displayed **ON** Shift of W-CS (workpiece coordinate system) is displayed

enableMirror 128712

Show mirror image in the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableMirror
Input:	**OFF** Mirroring is not displayed **ON** Mirroring is displayed

enableShiftMWCS 128711

Show shift of mW-CS in the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableShiftMWCS
Input:	**OFF** Shift of mW-CS (modified workpiece coordinate system) is not displayed **ON** Shift of the mW-CS (modified workpiece coordinate system) is displayed

enableRotation 128707

Show rotation in the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableRotation
Input:	**OFF** Rotation is not displayed **ON** Rotation is displayed

enableFeed 128708

Show feed rate the GPS dialog

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableFeed
Input:	**OFF** Feed rate is not displayed **ON** Freed rate is displayed

enableHwMCS 128709

M-CS coordinate system is selectable

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwMCS
Input:	**OFF** Coordinate system M-CS (machine coordinate system) cannot be selected **ON** Coordinate system M-CS (machine coordinate system) can be selected

enableHwWCS 128710

W-CS coordinate system is selectable

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwWCS
Input:	**OFF** Coordinate system W-CS (workpiece coordinate system) cannot be selected **ON** Coordinate system W-CS (workpiece coordinate system) can be selected

enableHwMWCS 128711

mW-CS coordinate system is selectable

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwMWCS
Input:	**OFF** Coordinate system mW-CS (modified workpiece coordinate system) cannot be selected **ON** Coordinate system mW-CS (modified workpiece coordinate system) can be selected

enableHwWPLCS 128712

WPL-CS coordinate system is selectable

Path:	System ▶ DisplaySettings ▶ CfgGlobalSettings ▶ enableHwWPLCS
Input:	**OFF** Coordinate system WPL-CS (working plane coordinate system) cannot be selected **ON** Coordinate system WPL-CS (working plane coordinate system) can be selected

CfgRemoteDesktop 133500

Settings for Remote Desktop connections

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop

Structural element:

connections 133501

List of Remote Desktop connections to be displayed

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ connections
Input:	Enter here the name of a RemoteFX connection from Remote Desktop Manager. max. 80 Characters

title 133502

Name of the OEM operating mode

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ title
Input:	Specifies the name of the OEM operating mode for display on the TNC and in the information bar.

dialogRes 133502.00501

Name of a text

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ title ▶ dialogRes
Input:	If the text is not meant to be language-sensitive, then it must be available with this name in a text resource file. In this case, enter the text at the “text” attribute. max. 40 Characters

text 133502.00502

Language-sensitive text

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ title ▶ text
Input:	This text is loaded from a text resource file and should not be changed here. If the text is not language-sensitive, then you must directly provide the text here. In this case, do not enter anything at the “dialogRes” attribute. max. 60 Characters

icon 133503

Path/name for optional icon graphic file

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ icon
Input:	max. 260 Characters

locations 133504

List with positions where this Remote Desktop connection is displayed

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ locations

Input:

opMode 133504.[Index].133401

Operating mode

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ locations ▶ [Index] ▶ opMode
Input:	max. 80 Characters

subOpMode 133504.[Index].133402

Optional submode for the operating mode specified in 'opMode'

Path:	System ▶ DisplaySettings ▶ CfgRemoteDesktop ▶ locations ▶ [Index] ▶ subOpMode
Input:	max. 80 Characters

ProbeSettings

CfgTT 122700

Configuration of the tool calibration

Path:	System ▶ ProbeSettings ▶ CfgTT
Structural element:	

spindleOrientMode 122704

M function for spindle orientation

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ spindleOrientMode
Input:	-1 to 999 ■ **-1** Spindle orientation directly by NC ■ **0** Function inactive ■ **1 to 999** Number of the M function for spindle orientation by the PLC
iTNC 530:	MP6560

probingRoutine 122705

Probing routine

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ probingRoutine
Input:	**MultiDirections** The probe contact is probed from several directions. **SingleDirection** The probe contact is probed from one direction.
iTNC 530:	6500 Bit 8

probingDirRadial 122706

Probing direction for tool radius measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ probingDirRadial
Input:	**X_Positive** **Y_Positive** **X_Negative** **Y_Negative** **Z_Positive** **Z_Negative**

iTNC 530:	MP6505

offsetToolAxis 122707

Distance from lower edge of tool to upper edge of stylus

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ offsetToolAxis
Input:	0.001 to 99.9999 [mm], max. 4 decimal places
iTNC 530:	MP6530

rapidFeed 122708

Rapid traverse in probing cycle for TT tool touch probe

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ rapidFeed
Input:	10 to 300000
iTNC 530:	MP6550

probingFeed 122709

Probing feed rate for tool measurement with non-rotating tool

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ probingFeed
Input:	1 to 3000
iTNC 530:	6520

probingFeedCalc 122710

Calculation of the probing feed rate

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ probingFeedCalc
Input:	**ConstantTolerance** Calculation of the probing feed rate with constant tolerance **VariableTolerance** Calculation of the probing feed rate with variable tolerance **ConstantFeed** Constant probing feed rate
iTNC 530:	6507

spindleSpeedCalc 122711

Speed determination method

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ spindleSpeedCalc
Input:	**Automatic** Automatically determine speed **MinSpindleSpeed** Always use minimum spindle speed

iTNC 530:	6500 Bit4

maxPeriphSpeedMeas 122712

Maximum permissible surface speed of the tool edge for radius measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ maxPeriphSpeedMeas
Input:	1 to 129 [m/min], max. 4 decimal places
iTNC 530:	6570

maxSpeed 122714

Maximum permissible speed during tool measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ maxSpeed
Input:	0 to 1000
iTNC 530:	6572

measureTolerance1 122715

Maximum permissible measuring error for tool measurement with rotating tool (first measurement error)

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ measureTolerance1
Input:	0.001 to 0.999 [mm], max. 3 decimal places
iTNC 530:	6510.0

measureTolerance2 122716

Maximum permissible measuring error for tool measurement with rotating tool (second measurement error)

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ measureTolerance2
Input:	0.001 to 0.999 [mm], max. 3 decimal places
iTNC 530:	6510.1

stopOnCheck 122717

NC stop during tool check

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ stopOnCheck
Input:	**TRUE** If the breakage tolerance is exceeded, the NC program is stopped and the error message **Tool broken** is displayed. **FALSE** The NC program is not stopped when the breakage tolerance is exceeded

iTNC 530:	6500 Bit5

stopOnMeasurement 122718

NC stop during tool measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ stopOnMeasurement
Input:	**TRUE** If the breakage tolerance is exceeded, the NC program is stopped and the error message **Touch point inaccessible** is displayed **FALSE** The NC program is not stopped when the breakage tolerance is exceeded
iTNC 530:	6500 Bit6

adaptToolTable 122719

Change the tool table during tool check and tool measurement

Path:	System ▶ ProbeSettings ▶ CfgTT ▶ [Key name of the TT] ▶ adaptToolTable
Input:	**AdaptNever** The tool table is not changed after tool check and tool measurement. **AdaptOnBoth** The tool table is changed after tool check and tool measurement. **AdaptOnMeasure** The tool table is changed after tool measurement.
iTNC 530:	6500 Bit11

CfgTTRoundStylus 114200

Configuration of a round stylus

Path:	System ▶ ProbeSettings ▶ CfgTTRoundStylus
Structural element:	

centerPos 114201

Coordinates of the TT tool touch probe stylus contact center with respect to the machine datum

Path:	System ▶ ProbeSettings ▶ CfgTTRoundStylus ▶ [Key name of the TT] ▶ centerPos
Input:	-99999.9999 to 99999.9999 [mm], max. 4 decimal places [0]: X coordinate [1]: Y coordinate [2]: Z coordinate

iTNC 530:	6580, 6581, 6582

safetyDistToolAx 114203

Safety clearance around the probe contact of the TT tool touch probe for pre-positioning in the tool-axis direction

Path:	System ▶ ProbeSettings ▶ CfgTTRoundStylus ▶ [Key name of the TT] ▶ safetyDistToolAx
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places
iTNC 530:	6540.0

safetyDistStylus 114204

Safety zone around the stylus for pre-positioning

Path:	System ▶ ProbeSettings ▶ CfgTTRoundStylus ▶ [Key name of the TT] ▶ safetyDistStylus
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places Safety clearance in the plane perpendicular to the tool axis
iTNC 530:	6540.1

CfgTTRectStylus 114300

Configuration of a rectangular stylus

Path:	System ▶ ProbeSettings ▶ CfgTTRectStylus
Structural element:	

centerPos 114313

Coordinates of the stylus center

Path:	System ▶ ProbeSettings ▶ CfgTTRectStylus ▶ [Key name of the TT] ▶ centerPos
Input:	Coordinates of the stylus center with respect to the machine datum -99999.9999 to 99999.9999 [mm], max. 4 decimal places
iTNC 530:	6580, 6581, 6582

safetyDistToolAx 114317

Set-up clearance above the stylus for pre-positioning

Path:	System ▶ ProbeSettings ▶ CfgTTRectStylus ▶ [Key name of the TT] ▶ safetyDistToolAx
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places Safety clearance in tool axis direction
iTNC 530:	6540.0

safetyDistStylus 114318

Safety zone around the stylus for pre-positioning

Path:	System ► ProbeSettings ► CfgTTRectStylus ► [Key name of the TT] ► safetyDistStylus
Input:	0.001 to 99999.9999 [mm], max. 4 decimal places
iTNC 530:	6540.1

ChannelSettings

CfgActivateKinem	204000
Active kinematics	
Path:	Channels ▶ ChannelSettings ▶ CfgActivateKinem
Structural element:	

kinemToActivate	204001
Kinematics to be activated / Active kinematics	
Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgActivateKinem ▶ kinemToActivate
Input:	max. 18 Characters Key names from Channels/Kinematics/**CfgKinComposModel**. Select the key name of the kinematics configuration to be activated. You can also see from this parameter which kinematics model is currently active.

kinemAtStartup	204002
The kinematics to be activated during control start-up	
Path:	Channels ▶ ChannelSettings ▶ CfgActivateKinem ▶ [Key name of the machining channel] ▶ kinemAtStartup
Input:	max. 18 Characters Here, you can enter the key name of a default kinematic model (from **CfgKinComposModel**) that is activated during every start-up of the control (regardless of which key name has been entered in CfgActivateKinem/kinemToActivate).
iTNC 530:	7506

CfgNcPgmBehaviour	200800
Specify the behavior of the NC program.	
Path:	Channels ▶ ChannelSettings ▶ CfgNcPgmBehaviour
Structural element:	

operatingTimeReset	200801
Reset the machining time when program starts.	
Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgNcPgmBehaviour ▶ operatingTimeReset
Input:	**TRUE** The machining time is reset at each program start. **FALSE**

The machining time is totaled.

plcSignalCycle		200803
PLC signal for the number of the pending machining cycle		
	Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgNcPgmBehaviour ▶ plcSignalCycle
	Input:	max. 500 Characters Name or number of a PLC word marker

CfgGeoTolerance		200900
Geometry tolerances		
	Path:	Channels ▶ ChannelSettings ▶ CfgGeoTolerance
	Structural element:	

circleDeviation		200901
Permissible deviation of the radius		
	Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoTolerance ▶ circleDeviation
	Input:	0.0001 to 0.016 [mm], max. 4 decimal places Enter the permissible deviation of the radius between the end point and starting point of the arc.
	iTNC 530:	7431

threadTolerance		200902
Permissible deviation in successive threads		
	Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoTolerance ▶ threadTolerance
	Input:	0.0001 to 999.9999 [mm], max. 9 decimal places Permissible deviation of the dynamically smoothed contour from the programmed thread contour.

moveBack		200903
Reserve for retraction movements		
	Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoTolerance ▶ moveBack
	Input:	0.0001 to 10 [mm], max. 9 decimal places

With this parameter you specify how far before a limit switch or a collision object a retraction movement should end.

CfgGeoCycle	201000
Configuration of the fixed cycles	
Path:	Channels ▶ ChannelSettings ▶ CfgGeoCycle
Structural element:	

pocketOverlap	201001
Overlap factor for pocket milling	
Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ pocketOverlap
Input:	0.001 to 1.414, max. 3 decimal places
iTNC 530:	7430

posAfterContPocket	201007
Traverse after machining the contour pocket	
Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ posAfterContPocket
Input:	**PosBeforeMachining** Move to the position from which the SL cycle was started. **ToolAxClearanceHeight** Move the tool axis to clearance height.
iTNC 530:	7420 Bit 4

displaySpindleErr	201002
Display the **Spindle is not rotating** error message if M3/M4 is not active	
Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ displaySpindleErr
Input:	**on** The error message is displayed **off** The error message is not displayed
iTNC 530:	7441

displayDepthErr	201003
Display the **Check the depth sign** error message	
Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ displayDepthErr

Input:	**on** Error message is displayed **off** Error message is not displayed
iTNC 530:	7441

apprDepCylWall 201004

Behavior when moving to wall of slot in the cylinder surface

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ apprDepCylWall
Input:	Defines the behavior for cutter movements to the wall of a slot in the cylinder surface when machining the slot with a milling cutter whose diameter is less than the slot diameter (e.g. Cycle 28). **LineNormal** The slot wall is approached and departed linearly. **CircleTangential** The slot wall is approached and departed tangentially; at the beginning and end of the slot a rounding arc with a diameter equal to the slot width is inserted.
iTNC 530:	7680 Bit 12

mStrobeOrient 201005

M function for spindle orientation in machining cycles

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ mStrobeOrient
Input:	-1 to 999 -1: Spindle orientation directly through the NC 0: Function not active 1 to 999: Number of the M function for spindle orientation through the PLC.
iTNC 530:	7442

suppressPlungeErr 201006

Do not show 'Plunging type is not possible' error message

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ suppressPlungeErr
Input:	**on** Error message is not displayed **off** Error message is displayed

restoreCoolant 201008

Behavior of M7 and M8 with Cycles 202 and 204

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ restoreCoolant
Input:	**TRUE** At the end of Cycles 202 and 204, the status of M7 and M8 is restored to that before the cycle call. **FALSE** At the end of Cycles 202 and 204, the status of M7 and M8 is not restored automatically.
iTNC 530:	7682

facMinFeedTurnSMAX 201009

Automatic feed rate reduction after attaining SMAX

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ facMinFeedTurnSMAX
Input:	1 to 100 [%], max. 1 decimal places If the maximum spindle speed SMAX is reached, the turning operation can no longer maintain the constant surface speed (VCONST:ON). The parameter determines whether the feed should be automatically reduced from this point to the center of rotation. Possible settings: ■ Factor = 100% (default value): Feed rate reduction deactivated. The feed rate from the turning cycle is used. ■ 0 < factor < 100%: Feed rate reduction is activated. The minimum feed rate Fmin is: Fmin = feed rate from turning cycle * factor

suppressResMatlWar 201010

Do not show "Residual material" warning

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgGeoCycle ▶ suppressResMatlWar
Input:	**Never** The "Residual material due to cutter geometry" warning is never suppressed **NCOnly** The “Residual material due to cutter geometry” warning is suppressed only in the Machine operating modes. **Always** The "Residual material due to cutter geometry" warning is always suppressed.

CfgStretchFilter 201100

Geometry filter for filtering out linear elements

Path:	Channels ▶ ChannelSettings ▶ CfgStretchFilter
Structural element:	

filterType 201101

Type of stretch filter

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgStretchFilter ▶ filterType
Input:	**Off** The filter is switched off. **ShortCut** Individual points on the polygon are omitted. If the connecting line from the middle point to the previous point or to the next point of three subsequent points on a polygon is within the tolerance band, the middle point will be omitted. **Average** The geometry filter smooths corners. This method moves the contour points in such a way that changes in direction are less distinct.

tolerance 201102

Maximum distance of the filtered to the unfiltered contour

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgStretchFilter ▶ tolerance
Input:	0 to 10 [mm], max. 5 decimal places Points that lie within this tolerance on the resulting new path will be filtered out. **0**: Stretch filter is off

maxLength 201103

Maximum length of the distance resulting from filtering

Path:	Channels ▶ ChannelSettings ▶ [Key name of the machining channel] ▶ CfgStretchFilter ▶ maxLength
Input:	0 to 1000 [mm], max. 3 decimal places **0**: Stretch filter is off

CfgThreadSpindle 113600

Path:	Channels ▶ ChannelSettings ▶ CfgThreadSpindle

Structural element:

sourceOverride 113603

Effective override potentiometer for feed rate during thread cutting

Path:	Channels ▶ ChannelSettings ▶ [Key name of machining channel] ▶ CfgThreadSpindle ▶ sourceOverride
Input:	The adjusted potentiometer is effective during thread cutting for shaft speed and feed rate. **FeedPotentiometer** (previous behavior of the TNC 640) During thread cutting, the potentiometer is effective for the feed rate knob. The potentiometer for the spindle speed knob is not active. **SpindlePotentiometer** (iTNC 530-compatible setting) During thread cutting, the potentiometer is effective for the spindle speed knob. The potentiometer for the feed rate override is disabled.

thrdWaitingTime 113601

Waiting time at reversal point in thread base

Path:	Channels ▶ ChannelSettings ▶ [Key name of machining channel] ▶ CfgThreadSpindle ▶ thrdWaitingTime
Input:	0 to 1 000 [s], max. 9 decimal places The spindle stops for this time at the bottom of the thread before starting again in the opposite direction of rotation.
iTNC 530:	7120.0

thrdPreSwitchTime 113602

Advanced switching time of spindle

Path:	Channels ▶ ChannelSettings ▶ [Key name of machining channel] ▶ CfgThreadSpindle ▶ thrdPreSwitchTime
Input:	0 to 1 000 [s], max. 9 decimal places The spindle is stopped at this time before reaching the bottom of the thread.
iTNC 530:	7120.1

limitSpindleSpeed 113604

Limit of spindle speed with Cycles 17, 207 and 18

Path:	Channels ▶ ChannelSettings ▶ [Key name of machining channel] ▶ CfgThreadSpindle ▶ limitSpindleSpeed
Input:	**TRUE**

	Spindle speed is limited so that it runs with constant speed approx. 1/3 of the time **FALSE** Limit not active
iTNC 530:	7160, Bit1

CfgEditorSettings

CfgEditorSettings	105400
Settings for the NC editor	
Path:	System ▶ EditorSettings ▶ CfgEditorSettings
Structural element:	

createBackup	105401
Generate a backup file *.bak	
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ createBackup
Input:	**TRUE** After you have edited a file, a backup file *.bak is automatically created before you save the file and exit the NC editor. **FALSE** No backup file *.bak is created. Select this setting if you do not need any backup files and want to save memory space.

deleteBack	105402
Behavior of the cursor after deletion of lines	
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ deleteBack
Input:	**TRUE** Behavior as with iTNC 530, the cursor is on the previous line **FALSE** The cursor is on the next line

cursorAround	105403
Behavior of the cursor on the first or last line	
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ cursorAround
Input:	**TRUE** Cursor jumps to the last or first line **FALSE** Cursor stands ready at the first or last line. Jumping not possible.

lineBreak	105404
Line break on NC blocks with more than one line	
Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ lineBreak
Input:	**ALL** Always break and display lines completely (multiline) **ACT** Only display the selected NC block completely (multiline)

	NO Only display all lines when the selected NC block is edited
iTNC 530:	7281.0

stdTNChelp 105405

Activate help graphics when entering cycle data

Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ stdTNChelp
Input:	**TRUE** Behavior as with iTNC 530: the help graphics are displayed automatically during cycle entry. **FALSE** The help graphics have to be called via the **CYCLE HELP ON/OFF** soft key.

toggleCyclDef 105406

Behavior of the cycle soft key row after a cycle entry

Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ toggleCyclDef
Input:	**TRUE** The cycle soft key row remains active after the cycle entry **FALSE** The cycle soft key row is hidden after the cycle entry

warningAtDEL 105407

Confirmation request when deleting an NC block.

Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ warningAtDEL
Input:	**TRUE** The confirmation request is displayed and must be confirmed by pressing DEL again. **FALSE** iTNC 530 behavior: The NC block is deleted without any request for confirmation.
iTNC 530:	7246

maxLineGeoSearch 105408

Line number up to which a test of the NC program is to be run.

Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ maxLineGeoSearch
Input:	The available value range depends on the performance of the control. For the TNC7, you can enter a value between 100 and 100 000. If the parameter is not part of the configuration, the minimal value 100 becomes effective.

iTNC 530:	7229

blockIncrement 105409

ISO programming: Block number increment

Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ blockIncrement
Input:	0 to 250
iTNC 530:	7220

useProgAxes 105410

Specify programmable axes

Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ useProgAxes
Input:	**TRUE** Use the axis configuration defined in the CfgChannelAxes/**progAxis** parameter (200301). On machines with traverse range switchover, the editor offers all axes that are included in at least one kinematic model of the machine. **FALSE** Use the default axis configuration XYZABCUVW.

enableStraightCut 105411

Allow or lock paraxial positioning blocks

Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ enableStraightCut
Input:	**TRUE** Paraxial positioning blocks are allowed. When an orange axis key is pressed, and in DIN/ISO when G07 is programmed, a paraxial positioning block is generated. **FALSE** Paraxial positioning blocks are locked. When an orange axis key is pressed, the TNC7 generates a straight-line interpolation (L block) instead of a paraxial positioning block.
iTNC 530:	7246

maxLineCommandSrch 105412

Number of lines for searching for identical syntax elements

Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ maxLineCommandSrch
Input:	The available value range depends on the performance of the control. For the TNC7, you can enter a value between 500 and 400 000. If the parameter is not part of the configuration, the initial value 20 000 becomes effective.

noParaxMode 105413

Allow/lock **FUNCTION PARAXCOMP/PARAXMODE** via soft key

Path:	System ▶ EditorSettings ▶ CfgEditorSettings ▶ noParaxMode
Input:	The input possibility over **SPEC FCT** → soft key **PROGRAM FUNCTIONS** → soft key **FUNCTION PARAX** is permitted or locked using **MP_noParaxMode**. (Machine parameter not part of the configuration means that the behavior is the same as FALSE) **FALSE** The **FUNCTION PARAX** soft key is displayed **TRUE** The **FUNCTION PARAX** soft key is not displayed

CfgPgmMgt

CfgPgmMgt 122100

Settings for the file management

Path:	System ▶ ProgramManager ▶ CfgPgmMgt
Structural element:	

dependentFiles 122101

Display of dependent files

Path:	System ▶ ProgramManager ▶ CfgPgmMgt ▶ dependentFiles
Input:	**AUTOMATIC** Dependent files are not displayed **MANUAL** Dependent files are displayed

CfgProgramCheck

CfgProgramCheck	129800
Settings for tool-usage files	
Path:	System ▶ ToolSettings ▶ CfgProgramCheck
Structural element:	

autoCheckTimeOut	129803
Timeout for creation of tool-usage files	
Path:	System ▶ ToolSettings ▶ CfgProgramCheck ▶ autoCheckTimeOut
Input:	Automatic creation of the tool-usage file is aborted if this time is exceeded. 1 to 500

autoCheckPrg	129801
Create tool-usage file for NC program	
Path:	System ▶ ToolSettings ▶ CfgProgramCheck ▶ autoCheckPrg
Input:	**NoAutoCreate** No tool-usage list will be generated upon selection of a program **OnProgSelectionIfNotExist** A tool-usage list will be generated upon program selection if the list does not already exist **OnProgSelectionIfNecessary** A tool-usage list will be generated upon program selection if the list does not already exist or if it contains obsolete data **OnProgSelectionAndModify** A tool usage list will be generated upon program selection if the list does not already exist, if it contains obsolete data, or if the NC program is modified afterwards using an editor

autoCheckPal	129802
Create pallet-usage files	
Path:	System ▶ ToolSettings ▶ CfgProgramCheck ▶ autoCheckPal
Input:	**NoAutoCreate** No tool-usage files will be generated upon pallet selection **OnProgSelectionIfNotExist** Upon pallet selection, those tool-usage lists that do not already exist will be generated **OnProgSelectionIfNecessary** Upon pallet selection, those tool-usage lists that do not already exist or that contain obsolete data will be generated **OnProgSelectionAndModify**

Upon pallet selection, tool-usage lists will be generated that do not already exist, that contain obsolete data, or whose NC programs are modified via the editor

CfgUserPath

CfgUserPath		102200
Paths for the end user		
Path:	System ▶ Paths ▶ CfgUserPath	
Structural element:		

ncDir		102201
List of drives and/or directories		
Path:	System ▶ Paths ▶ CfgUserPath ▶ ncDir	
Input:	max. 260 Characters This parameter is available only on the Windows programming stations of the TNC7. The parameter is not evaluated on a programming station with virtualization software (VBox) or on the TNC target system. The drives and/or directories entered here are visible in the file manager, provided that you have the required access rights. These paths may contain NC programs or tables. Possible entries are, for example: floppy-disk, HDR and CFR directories as well as network drives.	

fn16DefaultPath		102202
Default output path for the **FN16: F-PRINT** function in the Program Run operating modes		
Path:	System ▶ Paths ▶ CfgUserPath ▶ fn16DefaultPath	
Input:	max. 260 Characters Select the folder in the dialog window and confirm it with the **SELECT** soft key Default path for output with **FN 16: F-PRINT**. If no path is defined for the FN 16 function in the NC program, the output destination is in the directory specified here.	

fn16DefaultPathSim		102203
Default output path for the **FN16: F-PRINT** function in the Programming and Test Run operating modes		
Path:	System ▶ Paths ▶ CfgUserPath ▶ fn16DefaultPathSim	
Input:	max. 260 Characters Select the folder in the dialog window and confirm it with the **SELECT** soft key Default path for output with **FN 16: F-PRINT**. If no path is defined for the FN 16 function in the NC program, the output destination is in the directory specified here.	

serialInterfaceRS232

CfgSerialPorts	106600
Data record belonging to the serial port	
Path:	System ▶ Network ▶ Serial ▶ CfgSerialPorts
Structural element:	

activeRs232	106601
Enable the RS-232 interface in the program manager	
Path:	System ▶ Network ▶ Serial ▶ CfgSerialPorts ▶ activeRs232
Input:	**TRUE** The RS-232 interface is enabled in the program manager and shown as a drive icon (**RS232:**). **FALSE** The RS-232 interface cannot be accessed via the program manager.

baudRateLsv2	106606
Data transfer rate for LSV2 communication in baud	
Path:	System ▶ Network ▶ Serial ▶ CfgSerialPorts ▶ baudRateLsv2
Input:	Use a selection menu to define the transfer rate for the LSV2 communication. Minimum value is 110 baud, maximum value 115200 baud. **BAUD_110** **BAUD_150** **BAUD_300** **BAUD_600** **BAUD_1200** **BAUD_2400** **BAUD_4800** **BAUD_9600** **BAUD_19200** **BAUD_38400** **BAUD_57600** **BAUD_115200**

CfgSerialInterface	106700
Definition of data records for the serial ports	
Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface

Structural element:

baudRate 106701

Data transfer rate for communication in baud

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ baudRate
Input:	Use a selection menu to define the transfer rate for the data transmission. Minimum value is 110 baud, maximum value 115200 baud. **BAUD_110** **BAUD_150** **BAUD_300** **BAUD_600** **BAUD_1200** **BAUD_2400** **BAUD_4800** **BAUD_9600** **BAUD_19200** **BAUD_38400** **BAUD_57600** **BAUD_115200**
iTNC 530:	5040

protocol 106702

Communications protocol

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ protocol
Input:	**STANDARD** Standard data transfer. Data transferred line-by-line. **BLOCKWISE** Packet-based data transfer, ACK/NAK protocol. The control characters ACK (Acknowledge) and NAK (not Acknowledge) are used to control block-wise data transfer. **RAW_DATA** Data transferred without protocol. Transfer of characters without control characters. Protocol intended for transfer of data of the PLC.
iTNC 530:	5030

dataBits 106703

Data bits in each transferred character

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ dataBits

Input:	**7 bits** 7 data bits are transferred for each character transferred. **8 bits** 8 data bits are transferred for each character transferred.
iTNC 530:	5020 Bit0

parity 106704

Type of parity checking

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ parity
Input:	**NONE** No parity **EVEN** Even parity **ODD** Odd parity
iTNC 530:	5020 Bit4/5

stopBits 106705

Number of stop bits

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ stopBits
Input:	**1 stop bit** 1 stop bit is appended after each transferred character. **2 stop bits** 2 stop bits are appended after each transferred character.
iTNC 530:	5020 Bit6/7

flowControl 106706

Type of data-flow checking

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ flowControl
Input:	Configure here whether there is to be a data-flow check (handshake). **NONE** No data-flow check; handshake not active **RTS_CTS** Hardware handshake. Transmission stop through RTS active **XON_XOFF** Software handshake; Transfer stop by DC3 (XOFF) active
iTNC 530:	5020 Bit2/3

fileSystem 106707

File system for file operation via serial interface

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ fileSystem
Input:	**EXT** Minimum file system for external devices. Corresponds to the EXT1 and EXT2 modes of earlier TNC controls. Use these settings if you are using printers, punches, or non-HEIDENHAIN data transfer software. **FE1** Use this setting for communication with the external HEIDENHAIN FE 401 B or FE 401 floppy disk unit as of software 230626-03, or for communication with the "TNCserver" PC software from HEIDENHAIN.

bccAvoidCtrlChar 106708

Avoid control characters in the block check character (BCC)

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ bccAvoidCtrlChar
Input:	**TRUE** Ensures that the check sum does not correspond to a control character **FALSE** Function not active
iTNC 530:	5020 Bit1

rtsLow 106709

Idle state of the RTS line

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ rtsLow
Input:	**TRUE** The idle state of the RTS line is logical LOW **FALSE** The idle state of the RTS line is at logical HIGH
iTNC 530:	5020 Bit8

noEotAfterEtx 106710

Behavior after reception of an ETX control character

Path:	System ▶ Network ▶ Serial ▶ CfgSerialInterface ▶ [Key names of the interface parameters] ▶ noEotAfterEtx
Input:	**TRUE** No EOT control character is sent after reception of an ETX control character. **FALSE** The control sends an EOT control character after reception of an ETX control character.

iTNC 530:	5020 Bit9

Monitoring

CfgMonUser 129400

Monitoring settings for the user

Path:	System ▶ Monitoring ▶ ComponentMonitoring ▶ CfgMonUser
Structural element:	

enforceReaction 129401

The configured error reactions are enforced

Path:	System ▶ Monitoring ▶ ComponentMonitoring ▶ CfgMonUser ▶ enforceReaction
Input:	**TRUE** **FALSE**

showWarning 129402

Display warnings of monitoring tasks

Path:	System ▶ Monitoring ▶ ComponentMonitoring ▶ CfgMonUser ▶ showWarning
Input:	**TRUE** **FALSE**

CfgMonMbSection 133700

CfgMonMbSection defines monitoring tasks for a certain section of an NC program

Path:	System ▶ Monitoring ▶ ProcessMonitoring ▶ CfgMonMbSection
Structural element:	

tasks 133701

List of monitoring tasks to be performed

Path:	System ▶ Monitoring ▶ ProcessMonitoring ▶ CfgMonMbSection ▶ [Key name] ▶ tasks
Input:	

CfgMachineInfo

CfgMachineInfo		131700
General information of the machine operator		
Path:	System ▶ CfgMachineInfo	
Structural element:	Defines general information about this machine: ■ Settable by the user of the machine ■ Can be queried (e.g., via the OPC UA NC Server)	

machineNickname		131701
Custom name (nickname) of the machine		
Path:	System ▶ CfgMachineInfo ▶ machineNickname	
Input:	max. 64 Characters Machine designation freely selectable by the user.	

inventoryNumber		131702
Inventory number or ID		
Path:	System ▶ CfgMachineInfo ▶ inventoryNumber	
Input:	max. 64 Characters Internal inventory number of the operator's machine.	

image		131703
Photo or image of the machine		
Path:	System ▶ CfgMachineInfo ▶ image	
Input:	max. 260 Characters Path to an image file (*.jpg or *.png).	

location		131704
Machine location		
Path:	System ▶ CfgMachineInfo ▶ location	
Input:	max. 64 Characters	

department		131705
Department or division		
Path:	System ▶ CfgMachineInfo ▶ department	
Input:	max. 64 Characters	

responsibility		131706
Responsible for the machine		
Path:	System ▶ CfgMachineInfo ▶ responsibility	
Input:	max. 64 Characters	

Contact partner responsible for the machine, can be a person or a department.

contactEmail		131707
Contact email address		
Path:	System ▶ CfgMachineInfo ▶ contactEmail	
Input:	max. 64 Characters E-mail address of the responsible person or department.	

contactPhoneNumber		131708
Contact phone number		
Path:	System ▶ CfgMachineInfo ▶ contactPhoneNumber	
Input:	max. 32 Characters Telephone number of the responsible person or department.	

40.3 Preassigned error numbers for FN 14: ERROR

With the **FN 14** function you can issue error messages in the NC program.

Further information: "Output error messages with FN 14: ERROR", Page 1284

The following error messages are preassigned by HEIDENHAIN:

Error number	Text
1000	Spindle?
1001	Tool axis is missing
1002	Tool radius too small
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING FACTOR not permitted
1008	MIRROR IMAGE not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Input value incorrect
1012	Incorrect sign
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory input
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong rpm
1021	Radius comp. undefined
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive nesting
1026	Angle reference missing
1027	No fixed cycle defined
1028	Slot width too small
1029	Pocket too small
1030	Q202 not defined
1031	Q205 not defined
1032	Q218 must be greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted

Error number	Text
1035	Q220 too large
1036	Q222 must be greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be under 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted
1042	Traverse direction not defined
1043	No datum table active
1044	Position error: center in axis 1
1045	Position error: center in axis 2
1046	Hole diameter too small
1047	Hole diameter too large
1048	Stud diameter too small
1049	Stud diameter too large
1050	Pocket too small: rework axis 1
1051	Pocket too small: rework axis 2
1052	Pocket too large: scrap axis 1
1053	Pocket too large: scrap axis 2
1054	Stud too small: scrap axis 1
1055	Stud too small: scrap axis 2
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2
1058	TCHPROBE 425: length exceeds max
1059	TCHPROBE 425: length below min
1060	TCHPROBE 426: length exceeds max
1061	TCHPROBE 426: length below min
1062	TCHPROBE 430: diameter too large
1063	TCHPROBE 430: diameter too small
1064	No measuring axis defined
1065	Tool breakage tolerance exceeded
1066	Enter Q247 unequal to 0
1067	Enter Q247 greater than 5
1068	Datum table?
1069	Enter Q351 unequal to 0
1070	Thread depth too large
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active

Error number	Text
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter depth as negative
1078	Q303 in meas. cycle undefined!
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory meas. points
1082	Incorrect clearance height
1083	Contradictory plunge type
1084	This fixed cycle not allowed
1085	Line is write-protected
1086	Oversize greater than depth
1087	No point angle defined
1088	Contradictory data
1089	Slot position 0 not allowed
1090	Enter an infeed not equal to 0
1091	Switchover of Q399 not allowed
1092	Tool not defined
1093	Tool number not permitted
1094	Tool name not permitted
1095	Software option not active
1096	Kinematics cannot be restored
1097	Function not permitted
1098	Contradictory workpc. blank dim.
1099	Measuring position not allowed
1100	Kinematic access not possible
1101	Meas. pos. not in traverse range
1102	Preset compensation not possible
1103	Tool radius too large
1104	Plunging type is not possible
1105	Plunge angle incorrectly defined
1106	Angular length is undefined
1107	Slot width is too large
1108	Scaling factors not equal
1109	Tool data inconsistent
1110	MOVE not possible
1111	Presetting not allowed!
1112	Thread angle too small!

Error number	Text
1113	3-D ROT status is contradictory!
1114	Configuration is incomplete
1115	No turning tool is active
1116	Tool orientation is inconsistent
1117	Angle not possible!
1118	Radius too small!
1119	Thread runout too short!
1120	Contradictory meas. points
1121	Too many limits
1122	Machining strategy with limits not possible
1123	Machining direction not possible
1124	Check the thread pitch!
1125	Angle cannot be calculated
1126	Eccentric turning not possible
1127	No milling tool is active
1128	Insufficient length of cutting edge
1129	Gear definition is inconsistent or incomplete
1130	No finishing allowance provided
1131	Line does not exist in table
1132	Probing process not possible
1133	Coupling function not possible
1134	Machining cycle is not supported by this NC software
1135	Touch probe cycle is not supported by this NC software
1136	NC program aborted
1137	Touch probe data incomplete
1138	LAC function not possible
1139	Rounding radius or chamfer is too large!
1140	Axis angle not equal to tilt angle
1141	Character height not defined
1142	Excessive character height
1143	Tolerance error: Workpiece rework
1144	Tolerance error: Workpiece scrap
1145	Faulty dimension definition
1146	Illegal entry in compensation table
1147	Transformation not possible
1148	Tool spindle incorrectly configured
1149	Offset of the turning spindle unknown
1150	Global program settings are active
1151	Faulty configuration of OEM macros

Error number	Text
1152	The combination of programmed oversizes is not possible
1153	Measured value not captured
1154	Check the monitoring of the tolerance
1155	Hole is smaller than the stylus tip
1156	Preset cannot be set
1157	Alignment of a rotary table is not possible
1158	Alignment of rotary axes is not possible
1159	Infeed limited to length of cutting edge
1160	Machining depth defined as 0
1161	Tool type is unsuitable
1162	Finishing allowance not defined
1163	Machine datum could not be written
1164	Spindle for synchronization could not be ascertained
1165	Function is not possible in the active operating mode
1166	Oversize defined too large
1167	Number of teeth not defined
1168	Machining depth does not increase monotonously
1169	Infeed does not decrease monotonously
1170	Tool radius not defined correctly
1171	Mode for retraction to clearance height not possible
1172	Gear wheel definition incorrect
1173	Probing object contains different types of dimension definition
1174	Dimension definition contains impermissible characters
1175	Actual value in dimension definition faulty
1176	Starting point of hole too deep
1177	Dimension def.: Nominal value missing for manual pre-positioning
1178	A replacement tool is not available
1179	OEM macro is not defined
1180	Measurement not possible with auxiliary axis
1181	Start position not possible with modulo axis
1182	Function only possible if door is closed
1183	Number of possible records exceeded
1184	Inconsistent machining plane due to axis angle with basic rotation
1185	Transfer parameter contains an impermissible value
1186	Tooth width RCUTS is defined too large
1187	Usable length LU of the tool is too small
1188	The defined chamfer is too large

Error number	Text
1189	Chamfer angle cannot be machined with the active tool
1190	The allowances do not define any stock removal
1191	Spindle angle not unique

40.4 System data

40.4.1 List of FN functions

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Program information				
	10	3	-	Number of the active machining cycle
		6	-	Number of the most recently executed touch probe cycle –1 = None
		7	-	Type of calling NC program: –1 = None 0 = Visible NC program 1 = Cycle/macro, main program is visible 2 = Cycle/macro, there is no visible main program
		8	1	Unit of measure of the directly calling NC program (may also be a cycle). Return codes: 0 = mm 1 = inch –1 = there is no corresponding program
			2	Unit of measure of the NC program visible in the block display from which the current cycle was called directly or indirectly. Return codes: 0 = mm 1 = inch –1 = there is no corresponding program
		9	-	Within an M function macro: Number of the M function. Otherwise –1
		103	Q parameter number	Relevant within NC cycles; for inquiry as to whether the Q parameter given under IDX was explicitly stated in the associated CYCLE DEF.
		110	QS parameter number	Is there a file with the name QS(IDX)? 0 = No, 1 = Yes This function resolves relative file paths.
		111	QS parameter number	Is there a directory with the name QS(IDX)? 0 = no, 1 = Yes Only absolute directory paths are possible.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
System jump addresses				
	13	1	-	Label number or label name (string or QS) jumped to during M2/M30 instead of ending the current NC program. Value = 0: M2/M30 have the normal effect
		2	-	Label number or label name (string or QS) jumped to in the event of FN14: ERROR with the NC CANCEL reaction instead of aborting the NC program with an error message. The error number programmed in the FN14 command can be read under ID992 NR14. Value = 0: FN14 has the normal effect.
		3	-	Label number or label name (string or QS) jumped to in the event of an internal server error (SQL, PLC, CFG) or with erroneous file operations (FUNCTION FILECOPY, FUNCTION FILEMOVE, or FUNCTION FILEDELETE) instead of aborting the NC program with an error message. Value = 0: Error has the normal effect.
Indexed access to Q parameters				
	15	11	Q parameter number	Reads Q(IDX)
		12	QL parameter no.	Reads QL(IDX)
		13	QR parameter no.	Reads QR(IDX)
Machine status				
	20	1	-	Active tool number
		2	-	Prepared tool number
		3	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 8 = W
		4	-	Programmed spindle speed
		5	-	Active spindle condition -1 = spindle condition not defined 0 = M3 active 1 = M4 active 2 = M5 active after M3 3 = M5 active after M4
		7	-	Active gear range
		8	-	Active coolant status 0 = off, 1 = on
		9	-	Active feed rate

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	-	Index of prepared tool
		11	-	Index of active tool
		14	-	Number of active spindle
		20	-	Programmed cutting speed in turning operation
		21	-	Spindle mode in turning mode: 0 = constant speed 1 = constant cutting speed
		22	-	Coolant status M7: 0 = inactive, 1 = active
		23	-	Coolant status M8: 0 = inactive, 1 = active

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Channel data				
	25	1	-	Channel number
Cycle parameters				
	30	1	-	Set-up clearance
		2	-	Hole depth / milling depth
		3	-	Plunging depth
		4	-	Feed rate for plunging
		5	-	First side length of pocket
		6	-	Second side length of pocket
		7	-	First side length of slot
		8	-	Second side length of slot
		9	-	Radius of circular pocket
		10	-	Feed rate for milling
		11	-	Rotational direction of the milling path
		12	-	Dwell time
		13	-	Thread pitch for Cycles 17 and 18
		14	-	Finishing allowance
		15	-	Roughing angle
		21	-	Probing angle
		22	-	Probing path
		23	-	Probing feed rate
		49	-	HSC mode (Cycle 32 Tolerance)
		50	-	Tolerance for rotary axes (Cycle 32 Tolerance)
		52	Q parameter number	Type of transfer parameter for user cycles: –1: Cycle parameter not programmed in CYCL DEF 0: Cycle parameter numerically programmed in CYCL DEF (Q parameter) 1: Cycle parameter programmed as string in CYCL DEF (Q parameter)
		60	-	Clearance height (touch probe cycles 30 to 33)
		61	-	Inspection (touch probe cycles 30 to 33)
		62	-	Cutting edge measurement (touch probe cycles 30 to 33)
		63	-	Q parameter number for the result (touch probe cycles 30 to 33)
		64	-	Q parameter type for the result (touch probe cycles 30 to 33) 1 = Q, 2 = QL, 3 = QR

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		70	-	Multiplier for feed rate (cycles 17 and 18)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Modal status				
	35	1	-	Dimensions: 0 = absolute (G90) 1 = incremental (G91)
		2	-	Radius compensation: 0 = R0 1 = RR/RL 10 = Face milling 11 = Peripheral milling
Data for SQL tables				
	40	1	-	Result code for the last SQL command. If the last result code was 1 (=error), the error code is transferred as the return code.
Data from the tool table				
	50	1	Tool no.	Tool length L
		2	Tool no.	Tool radius R
		3	Tool no.	Tool radius R2
		4	Tool no.	Oversize for tool length DL
		5	Tool no.	Tool radius oversize DR
		6	Tool no.	Tool radius oversize DR2
		7	Tool no.	Tool locked TL 0 = not locked, 1 = locked
		8	Tool no.	Number of the replacement tool RT
		9	Tool no.	Maximum tool age TIME1
		10	Tool no.	Maximum tool age TIME2
		11	Tool no.	Current tool age CUR.TIME
		12	Tool no.	PLC status
		13	Tool no.	Maximum tooth length LCUTS
		14	Tool no.	Maximum plunge angle ANGLE
		15	Tool no.	TT: Number of tool teeth CUT
		16	Tool no.	TT: Wear tolerance for length, LTOL
		17	Tool no.	TT: Wear tolerance for radius, RTOL
		18	Tool no.	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	Tool no.	TT: Offset in plane R-OFFS R = 99999.9999
		20	Tool no.	TT: Offset in length L-OFFS
		21	Tool no.	TT: Breakage tolerance for length, LBREAK
		22	Tool no.	TT: Breakage tolerance for radius, RBREAK

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		28	Tool no.	Maximum speed NMAX
		32	Tool no.	Point angle TANGLE
		34	Tool no.	LIFTOFF allowed (0 = No, 1 = Yes)
		35	Tool no.	Wear tolerance for radius R2TOL
		36	Tool no.	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	Tool no.	Corresponding line in the touch-probe table
		38	Tool no.	Timestamp of last use
		39	Tool no.	ACC
		40	Tool no.	Pitch for thread cycles
		41	Tool no.	AFC: reference load
		42	Tool no.	AFC: overload early warning
		43	Tool no.	AFC: overload NC stop
		44	Tool no.	Exceeding the tool life
		45	Tool no.	Front-face width of indexable insert (RCUTS)
		46	Tool no.	Usable length of the milling cutter
		47	Tool no.	Neck radius of the milling cutter (RN)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Data from the pocket table				
	51	1	Pocket number	Tool number
		2	Pocket number	0 = no special tool 1 = special tool
		3	Pocket number	0 = no fixed pocket 1 = fixed pocket
		4	Pocket number	0 = pocket not locked 1 = pocket locked
		5	Pocket number	PLC status
Determine the tool pocket				
	52	1	Tool no.	Pocket number
		2	Tool no.	Tool magazine number
File information				
	56	1	-	Number of lines of the tool table
		2	-	Number of lines of the active datum table
		4	-	Number of lines in a freely definable table that has been opened with FN26: TABOPEN
Tool data for T and S strobes				
	57	1	T code	Tool number IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		2	T code	Tool index IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		5	-	Spindle speed IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
Values programmed in TOOL CALL				
	60	1	-	Tool number T
		2	-	Active tool axis 0 = X 1 = Y 2 = Z 6 = U 7 = V 8 = W
		3	-	Spindle speed S
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Automatic TOOL CALL 0 = Yes, 1 = No

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		7	-	Tool radius oversize DR2
		8	-	Tool index
		9	-	Active feed rate
		10	-	Cutting speed [mm/min]
Values programmed in TOOL DEF				
	61	0	Tool no.	Read the number of the tool change sequence: 0 = Tool already in spindle, 1 = Change between external tools, 2 = Change from internal to external tool, 3 = Change from special tool to external tool, 4 = Load external tool, 5 = Change from external to internal tool, 6 = Change from internal to internal tool, 7 = Change from special tool to internal tool, 8 = Load internal tool, 9 = Change from external tool to special tool, 10 = Change from special tool to internal tool, 11 = Change from special tool to special tool, 12 = Load special tool, 13 = Unload external tool, 14 = Unload internal tool, 15 = Unload special tool
		1	-	Tool number T
		2	-	Length
		3	-	Radius
		4	-	Index
		5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed with FUNCTION TURNDATA				
	62	1	-	Tool length oversize DXL
		2	-	Tool length oversize DYL
		3	-	Tool length oversize DZL
		4	-	Cutting radius oversize DRS
Values for LAC and VSC				
	71	0	0	Index of the NC axis for which the LAC weighing run will be performed or was last performed (X to W = 1 to 9)
			2	Total inertia determined by the LAC weighing run in [kgm²] (with A/B/C rotary axes) or total mass in [kg] (with X/Y/Z linear axes)
		1	0	Cycle 957 Retraction from thread
Freely available memory area for OEM cycles				
	72	0-39	0 to 30	Freely available memory area for OEM cycles. The values are only reset by the control during a control reboot (= 0). With “Cancel,” the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Freely available memory area for user cycles				
	73	0-39	0 to 30	Freely available memory area for user cycles The values are only reset by the control during a control reboot (= 0). With “Cancel,” the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Read minimum and maximum spindle speed				
	90	1	Spindle ID	Minimum spindle speed of the lowest gear stage. If no gear stages are configured, CfgFeedLimits/minFeed of the first parameter set of the spindle is evaluated. Index 99 = active spindle
		2	Spindle ID	Maximum spindle speed from the highest gear stage. If no gear stages are configured, CfgFeedLimits/maxFeed of the first parameter set of the spindle is evaluated. Index 99 = active spindle

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Tool compensation				
	200	1	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active radius
		2	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active length
		3	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Rounding radius R2
		6	Tool no.	Tool length Index 0= active tool
Coordinate transformations				
	210	1	-	Basic rotation (manual)
		2	-	Programmed rotation
		3	-	Active mirror axis. Bits 0 to 2 and 6 to 8: Axes X, Y, Z and U, V, W
		4	Axis	Active scaling factor Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	Rotary axis	3D-ROT Index: 1 - 3 (A, B, C)
		6	-	Tilt working plane in Program Run operating modes 0 = Not active –1 = Active
		7	-	Tilt working plane in Manual operating modes 0 = Not active –1 = Active
		8	QL parameter no.	Angle of misalignment between spindle and tilted coordinate system. Projects the angle specified in the QL parameter from the input coordinate

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				system to the tool coordinate system. If IDX is omitted, the angle 0 is used for projection.
		10	-	Type of definition of the active tilt: 0 = no tilt—is returned if, both in **Manual Operation** and in the automatic modes, no tilt is active. 1 = axial 2 = spatial angle
		11	-	Coordinate system for manual movements: 0 = Machine coordinate system **M-CS** 1 = Working plane coordinate system **WPL-CS** 2 = Tool coordinate system **T-CS** 4 = Workpiece coordinate system **W-CS**
		12	Axis	Correction in working plane coordinate system **WPL-CS** (FUNCTION TURNDATA CORR WPL or FUNCTION CORRDATA WPL) Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Active coordinate system				
	211	–	-	1 = input system (default) 2 = REF system 3 = tool change system
Special transformations in turning mode				
	215	1	-	Angle for the precession of the input system in the XY plane in turning mode. To reset the transformation the value 0 must be entered for the angle. This transformation is used in connection with Cycle 800 (parameter Q497).
		3	1-3	Reading out of the spatial angle written with NR2 Index: 1 - 3 (rotA, rotB, rotC)
Current datum shift				
	220	2	Axis	Current datum shift in [mm] Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Read the difference between reference point and preset. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		4	Axis	Read values for OEM offset.. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Traverse range				
	230	2	Axis	Negative software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Positive software limit switches Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		5	-	Software limit switch on or off: 0 = on, 1 = off For modulo axes, either both the upper and lower limits or no limit at all must be set.
Read the nominal position in the REF system				
	240	1	Axis	Current nominal position in the REF system
Read the nominal position in the REF system, including offsets (handwheel, etc.)				
	241	1	Axis	Current nominal position in the REF system
Read the current position in the active coordinate system				
	270	1	Axis	Current nominal position in the input system When called while tool radius compensation is active, the function supplies the uncompensated positions for the principal axes X, Y, and Z. If the function is called for a rotary axis and tool radius

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				compensation is active, an error message is issued. Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
Read the current position in the active coordinate system, including offsets (handwheel, etc.)				
	271	1	Axis	Current nominal position in the input system
Read information to M128				
	280	1	-	M128 active: –1 = Yes, 0 = No
		3	-	Condition of TCPM after Q No.: Q No. + 0: TCPM active, 0 = no, 1 = yes Q No. + 1: AXIS, 0 = POS, 1 = SPAT Q No. + 2: PATHCTRL, 0 = AXIS, 1 = VECTOR Q No. + 3: Feed rate, 0 = F TCP, 1 = F CONT
Machine kinematics				
	290	5	-	0: Temperature compensation not active 1: Temperature compensation active
		7	-	KinematicsComp: 0: Compensations by KinematicsComp not active 1: Compensations by KinematicsComp active
		10	-	Index of the machine kinematics from Channels/ChannelSettings/CfgKin-List/kinCompositeModels programmed in FUNCTION MODE MILL or FUNCTION MODE TURN –1 = Not programmed.
Read data of the machine kinematics				
	295	1	QS parameter no.	Read the axis names of the active 3-axis kinematics. The axis names are written according to QS(IDX), QS(IDX+1), and QS(IDX+2). 0 = Operation successful
		2	0	Is FACING HEAD POS function active? 1 = Yes, 0 = No
		4	Rotary axis	Read whether the defined rotary axis participates in the kinematic calculation. 1 = Yes, 0 = No (A rotary axis can be excluded from the kinematics calculating using M138.) Index: 4, 5, 6 (A, B, C)
		5	Secondary axis	Read whether the given secondary axis is used in the kinematics model. –1 = Axis not in the kinematics model 0 = Axis is not included in the kinematics calculation:

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		6	Axis	Angle head: Displacement vector in the basic coordinate system B-CS through angle head Index: 1, 2, 3 (X, Y, Z)
		7	Axis	Angle head: Direction vector of the tool in the basic coordinate system B-CS Index: 1, 2, 3 (X, Y, Z)
		10	Axis	Determine programmable axes. Determine the axis ID associated with the specified axis index (index from CfgAxis/axisList). Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		11	Axis ID	Determine programmable axes. Determine the index of the axis (X = 1, Y = 2, ...) for the specified axis ID Index: Axis ID (index from CfgAxis/axisList)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Modify the geometrical behavior				
	310	20	Axis	Diameter programming: –1 = on, 0 = off
		126	-	M126: –1 = on, 0 = off
Current system time				
	320	1	0	System time in seconds that have elapsed since 01.01.1970, 00:00:00 (real time).
			1	System time in seconds that have elapsed since 01.01.1970, 00:00:00 (look-ahead calculation).
		3	-	Read the processing time of the current NC program.
Formatting of system time				
	321	0	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY hh:mm:ss
		1	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm:ss
		2	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm
		3	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY h:mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		4	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm:ss
		5	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm
		6	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD h:mm
		7	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD h:mm
		8	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY
		9	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY
		11	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD
		12	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD
		13	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: hh:mm:ss
		14	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm:ss
		15	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		16	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY hh:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY hh:mm
		20	0	The current calendar week number according to ISO 8601 (real time)
			1	The current calendar week number according to ISO 8601 (look-ahead calculation)
Global Program Settings (GPS): Global activation status				
	330	0	-	0 = No GPS setting is active 1 = Any GPS setting is active
Global Program Settings (GPS): Individual activation status				
	331	0	-	0 = No GPS setting is active 1 = Any GPS setting is active
		1	-	GPS: Basic rotation 0 = Off, 1 = On
		3	Axis	GPS: Mirroring 0 = Off, 1 = On Index: 1 - 6 (X, Y, Z, A, B, C)
		4	-	GPS: Shift in the modified workpiece system 0 = Off, 1 = On
		5	-	GPS: Rotation in input system 0 = Off, 1 = On
		6	-	GPS: Feed rate factor 0 = Off, 1 = On
		8	-	GPS: Handwheel superimpositioning 0 = Off, 1 = On
		10	-	GPS: Virtual tool axis VT 0 = Off, 1 = On
		15	-	GPS: Selection of the handwheel coordinate system 0 = Machine coordinate system M-CS 1 = Workpiece coordinate system W-CS 2 = Modified workpiece coordinate system mW-CS 3 = Working plane coordinate system WPL-CS
		16	-	GPS: Shift in the workpiece system 0 = Off, 1 = On
		17	-	GPS: Axis offset 0 = Off, 1 = On

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Global Program Settings (GPS)				
	332	1	-	GPS: Angle of a basic rotation
		3	Axis	GPS: Mirroring 0 = Not mirrored, 1 = Mirrored Index: 1 - 6 (X, Y, Z, A, B, C)
		4	Axis	GPS: Shift in the modified workpiece coordinate system mW-CS Index: 1 - 6 (X, Y, Z, A, B, C)
		5	-	GPS: Angle of rotation in input coordinate system I-CS
		6	-	GPS: Feed rate factor
		8	Axis	GPS: Handwheel superimpositioning Maximum value Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		9	Axis	GPS: Value for handwheel superimposi-tioning Index: 1 - 10 (X, Y, Z, A, B, C, U, V, W, VT)
		16	Axis	GPS: Shift in the workpiece coordinate system W-CS Index: 1 - 3 (X, Y, Z)
		17	Axis	GPS: Axis offset Index: 4 - 6 (A, B, C)
TS touch trigger probe				
	350	50	1	Touch probe type: 0: TS120, 1: TS220, 2: TS440, 3: TS630, 4: TS632, 5: TS640, 6: TS444, 7: TS740
			2	Line in the touch-probe table
		51	-	Effective length
		52	1	Effective radius of the stylus tip
			2	Rounding radius
		53	1	Center offset (reference axis)
			2	Center offset (minor axis)
		54	-	Spindle-orientation angle in degrees (center offset)
		55	1	Rapid traverse
			2	Measuring feed rate
			3	Feed rate for pre-positioning: FMAX_PROBE or FMAX_MACHINE
		56	1	Maximum measuring range
			2	Set-up clearance
		57	1	Spindle orientation possible 0=No, 1=Yes
			2	Angle of spindle orientation in degrees

Group name	Group number ID...	System data number NO....	Index IDX...	Description
TT tool touch probe for tool measurement				
	350	70	1	TT: Touch probe type
			2	TT: Line in the tool touch probe table
		71	1/2/3	TT: Touch probe center (REF system)
		72	-	TT: Touch probe radius
		75	1	TT: Rapid traverse
			2	TT: Measuring feed rate with stationary spindle
			3	TT: Measuring feed rate with rotating spindle
		76	1	TT: Maximum probing path
			2	TT: Safety clearance for linear measurement
			3	TT: Safety clearance for radius measurement
			4	TT: Distance from the lower edge of the cutter to the upper edge of the stylus
		77	-	TT: Spindle speed
		78	-	TT: Probing direction
		79	-	TT: Activate radio transmission
			-	TT: Stop probing movement upon stylus deflection

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Preset from touch probe cycle (probing results)				
	360	1	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (input coordinate system). Compensations: length, radius, and center offset
		2	Axis	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (machine coordinate system, only axes from the active 3-D kinematics are allowed as index). Compensation: only center offset
		3	Coordinate	Result of measurement in the input system of touch probe Cycles 0 and 1. The measurement result is read out in the form of coordinates. Compensation: only center offset
		4	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (workpiece coordinate system). The measurement result is read in the form of coordinates. Compensation: only center offset
		5	Axis	Axis values, not compensated
		6	Coordinate / axis	Readout of the measurement results in the form of coordinates / axis values in the input system from probing operations. Compensation: only length
		10	-	Oriented spindle stop
		11	-	Error status of probing: 0: Probing was successful –1: Touch point not reached –2: Touch probe already deflected at the start of the probing process

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read values from or write values to the active datum table				
	500	Row number	Column	Read values
Read values from or write values to the preset table (basic transformation)				
	507	Row number	1-6	Read values
Read axis offsets from or write axis offsets to the preset table				
	508	Row number	1-9	Read values
Data for pallet machining				
	510	1	-	Active line
		2	-	Current pallet number. Read value of the NAME column of the last PAL-type entry. If the column is empty or does not contain a numerical value, a value of –1 is returned.
		3	-	Active row of the pallet table.
		4	-	Last line of the NC program for the current pallet.
		5	Axis	Tool-oriented editing: Clearance height is programmed: 0 = No, 1 = Yes Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		6	Axis	Tool-oriented editing: Clearance height The value is invalid if ID510 NR5 returns the value 0 with the corresponding IDX. Index: 1 - 9 (X, Y, Z, A, B, C, U, V, W)
		10	-	Row number up to which the pallet table is to be searched during block scan.
		20	-	Type of pallet editing? 0 = Workpiece-oriented 1 = Tool oriented
		21	-	Automatic continuation after NC error: 0 = Locked 1 = Active 10 = Abort continuation 11 = Continuation with the rows in the pallet table that would have been executed next if not for the NC error 12 = Continuation with the row in the pallet table in which the NC error arose 13 = Continuation with the next pallet

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read data from the point table				
	520	Row number	10	Read value from active point table.
			11	Read value from active point table.
			1-3 X/Y/Z	Read value from active point table.
Read or write the active preset				
	530	1	-	Number of the active preset in the active preset table.
Active pallet preset				
	540	1	-	Number of the active pallet preset. Returns the number of the active preset. If no pallet preset is active, the function returns the value –1.
		2	-	Number of the active pallet preset. As with NR1.
Values for the basic transformation of the pallet preset				
	547	Row number	Axis	Read values of the basic transformation from the pallet preset table. Index: 1 to 6 (X, Y, Z, SPA, SPB, SPC)
Axis offsets from the pallet preset table				
	548	Row number	Offset	Read values of the axis offsets from the pallet preset table. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
OEM offset				
	558	Row number	Offset	Read values for OEM offset. Index: 1 - 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Read and write the machine status				
	590	2	1-30	Freely available; not deleted during program selection.
		3	1-30	Freely available; not deleted during a power failure (persistent storage).
Read/write look-ahead parameter of a single axis (at machine level)				
	610	1	-	Minimum feed rate (**MP_minPathFeed**) in mm/min
		2	-	Minimum feed rate at corners (**MP_minCornerFeed**) in mm/min
		3	-	Feed-rate limit for high speeds (**MP_maxG1Feed**) in mm/min
		4	-	Max. jerk at low speeds (**MP_maxPathJerk**) in m/s^3
		5	-	Max. jerk at high speeds (**MP_maxPathJerkHi**) in m/s^3
		6	-	Tolerance at low speeds (**MP_pathTolerance**) in mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		7	-	Tolerance at high speeds (**MP_pathToleranceHi**) in mm
		8	-	Max. derivative of jerk (**MP_maxPathYank**) in m/s^4
		9	-	Tolerance factor for curve machining (**MP_curveTolFactor**)
		10	-	Factor for max. permissible jerk at curvature changes (**MP_curveJerkFactor**)
		11	-	Maximum jerk with probing movements (**MP_pathMeasJerk**)
		12	-	Angle tolerance for machining feed rate (**MP_angleTolerance**)
		13	-	Angle tolerance for rapid traverse (**MP_angleToleranceHi**)
		14	-	Max. corner angle for polygons (**MP_maxPolyAngle**)
		18	-	Radial acceleration with machining feed rate (**MP_maxTransAcc**)
		19	-	Radial acceleration with rapid traverse (**MP_maxTransAccHi**)
		20	Index of physical axis	Max. feed rate (**MP_maxFeed**) in mm/min
		21	Index of physical axis	Max. acceleration (**MP_maxAcceleration**) in m/s^2
		22	Index of physical axis	Maximum transition jerk of the axis in rapid traverse (**MP_axTransJerkHi**) in m/s^2
		23	Index of physical axis	Maximum transition jerk of the axis during machining free rate (**MP_axTransJerk**) in m/s^3
		24	Index of physical axis	Acceleration feedforward control (**MP_compAcc**)
		25	Index of physical axis	Axis-specific jerk at low speeds (**MP_axPathJerk**) in m/s^3
		26	Index of physical axis	Axis-specific jerk at high speeds (**MP_axPathJerkHi**) in m/s^3
		27	Index of physical axis	More precise tolerance examination in corners (**MP_reduceCornerFeed**) 0 = deactivated, 1 = activated
		28	Index of physical axis	DCM: Maximum tolerance for linear axes in mm (**MP_maxLinearTolerance**)
		29	Index of physical axis	DCM: Maximum angle tolerance in [°] (**MP_maxAngleTolerance**)
		30	Index of physical axis	Tolerance monitoring for successive threads (**MP_threadTolerance**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		31	Index of physical axis	Form (**MP_shape**) of the **axisCutterLoc** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		32	Index of physical axis	Frequency (**MP_frequency**) of the **axisCutterLoc** filter in Hz
		33	Index of physical axis	Form (**MP_shape**) of the **axisPosition** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		34	Index of physical axis	Frequency (**MP_frequency**) of the **axisPosition** filter in Hz
		35	Index of physical axis	Order of the filter for **Manual** operating mode (**MP_manualFilterOrder**)
		36	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisCutterLoc** filter
		37	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisPosition** filter
		38	Index of physical axis	Axis-specific jerk for probing movements (**MP_axMeasJerk**)
		39	Index of physical axis	Weighting of the filter error for calculating filter deviation (**MP_axFilterErrWeight**)
		40	Index of physical axis	Maximum filter length of position filter (**MP_maxHscOrder**)
		41	Index of physical axis	Maximum filter length of CLP filter (**MP_maxHscOrder**)
		42	-	Maximum feed rate of the axis at machining feed rate (**MP_maxWorkFeed**)
		43	-	Maximum path acceleration at machining feed rate (**MP_maxPathAcc**)
		44	-	Maximum path acceleration at rapid traverse (**MP_maxPathAccHi**)
		45	-	Shape of the smoothing filter (**CfgSmoothingFilter/shape**) 0 = Off 1 = Average 2 = Triangle
		46	-	Order of smoothing filter (only odd-numbered values) (**CfgSmoothingFilter/order**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		47	-	Type of acceleration profile (**CfgLaPath/profileType**) 0 = Bellshaped 1 = Trapezoidal 2 = Advanced Trapezoidal
		48	-	Type of acceleration profile for rapid traverse (**CfgLaPath/profileTypeHi**) 0 = Bellshaped 1 = Trapezoidal 2 = Advanced Trapezoidal
		51	Index of physical axis	Compensation of following error in the jerk phase (**MP_IpcJerkFact**)
		52	Index of physical axis	kv factor of the position controller in 1/s (**MP_kvFactor**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Measure the maximum utilization of an axis				
	621	0	Index of physical axis	Conclude measurement of the dynamic load and save the result in the specified Q parameter.
Read SIK contents				
	630	0	Option no.	You can explicitly determine whether the SIK option given under **IDX** has been set or not. 1 = option is enabled 0 = option is not enabled
		1	-	You can determine whether a Feature Content Level (for upgrade functions) is set, and which one. –1 = No FCL is set <No.> = FCL that is set
		2	-	Read serial number of the SIK -1 = No valid SIK in the system
		10	-	Define the type of control: 0 = iTNC 530 1 = NCK-based control (TNC7, TNC 640, TNC 620, TNC 320, TNC 128, PNC 610, ...)
General data of the grinding wheel				
	780	2	-	Width
		3	-	Overhang
		4	-	Alpha angle (optional)
		5	-	Gamma angle (optional)
		6	-	Depth (optional)
		7	-	Rounding radius at the "Further" edge (optional)
		8	-	Rounding radius at the "Nearer" edge (optional)
		9	-	Rounding radius at the "Nearest" edge (optional)
		10	-	Active edge:
		11	-	Type of grinding wheel (straight / angular)
		12	-	External or internal wheel?
		13	-	Compensation angle of the B axis (with respect to the base angle of the location)
		14	-	Type of angular wheel
		15	-	Total length of the grinding wheel
		16	-	Length of the inner edge of the grinding wheel
		17	-	Minimum wheel diameter (wear limit)
		18	-	Minimum wheel width (wear limit)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		19	-	Tool number
		20	-	Cutting speed
		21	-	Maximum permissible cutting speed
		27	-	Wheel basic type: with relief cut
		28	-	Relief cut on the outside
		29	-	Relief cut on the inside
		30	-	Definition status
		31	-	Radius compensation
		32	-	Compensation of total length
		33	-	Compensation of overhang
		34	-	Compensation for the length to the innermost edge
		35	-	Radius of the shaft of the grinding wheel
		36	-	Initial dressing performed?
		37	-	Dresser location for initial dressing
		38	-	Dresser tool for initial dressing
		39	-	Has the grinding wheel been measured?
		51	-	Dresser tool for dressing on the diameter
		52	-	Dresser tool for dressing on the outer edge
		53	-	Dresser tool for dressing on the inner edge
		54	-	Dressing of the diameter according to the number of calls
		55	-	Dressing of the outer edge according to the number of calls
		56	-	Dressing of the inner edge according to the number of calls
		57	-	Dressing counter of the diameter
		58	-	Dressing counter of the outer edge
		59	-	Dressing counter of the inner edge
		101	-	Radius of grinding wheel

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Detailed geometry (contour) of the grinding wheel				
	783	1	1	Chamfer width of the outer side of the wheel
			2	Chamfer width of the inner side of the wheel
		2	1	Chamfer angle of the outer side of the wheel
			2	Chamfer angle of the inner side of the wheel
		3	1	Corner radius of the outer side of the wheel
			2	Corner radius of the inner side of the wheel
		4	1	Side length of the outer side of the wheel
			2	Side length of the inner side of the wheel
		5	1	Relief length of the outer side of the wheel
			2	Relief length of the inner side of the wheel
		6	1	Relief angle of the outer side of the wheel
			2	Relief angle of the inner side of the wheel
		7	1	Recess length of the outer side of the wheel
			2	Recess length of the inner side of the wheel
		8	1	Departing radius of the outer side of the wheel
			2	Departing radius of the inner side of the wheel
		9	1	Total depth on the outside
			2	Total depth on the inside

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read Functional Safety (FS) information				
	820	1	-	FS limitations: 0 = No Functional Safety (FS) 1 = Guard door open (SOM1) 2 = Guard door open (SOM2) 3 = Guard door open (SOM3) 4 = Guard door open (SOM4) 5 = All guard doors closed
Write data for unbalance monitoring				
	850	10	-	Activate and deactivate unbalance monitoring 0 = unbalance monitoring not active 1 = unbalance monitoring active
Counter				
	920	1	-	Planned workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		2	-	Already machined workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		12	-	Workpieces still to be machined. In **Test Run** operating mode the counter generally generates the value 0.
Read and write data of current tool				
	950	1	-	Tool length L
		2	-	Tool radius R
		3	-	Tool radius R2
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Tool radius oversize DR2
		7	-	Tool locked TL 0 = not locked, 1 = locked
		8	-	Number of the replacement tool RT
		9	-	Maximum tool age TIME1
		10	-	Maximum tool age TIME2 at TOOL CALL
		11	-	Current tool age CUR.TIME
		12	-	PLC status
		13	-	Tooth length in the tool axis LCUTS
		14	-	Maximum plunge angle ANGLE
		15	-	TT: Number of tool teeth CUT
		16	-	TT: Wear tolerance for length LTOL
		17	-	TT: Wear tolerance for radius RTOL

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		18	-	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	-	TT: Offset in plane R-OFFS R = 99999.9999
		20	-	TT: Offset in length L-OFFS
		21	-	TT: Break tolerance for length LBREAK
		22	-	TT: Break tolerance for radius RBREAK
		28	-	Maximum spindle speed [rpm] NMAX
		32	-	Point angle TANGLE
		34	-	LIFTOFF allowed (0 = No, 1 = Yes)
		35	-	Wear tolerance for radius R2TOL
		36	-	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	-	Corresponding line in the touch-probe table
		38	-	Timestamp of last use
		39	-	ACC
		40	-	Pitch for thread cycles
		41	-	AFC: reference load
		42	-	AFC: overload early warning
		43	-	AFC: overload NC stop
		44	-	Exceeding the tool life
		45	-	Front-face width of indexable insert (RCUTS)
		46	-	Usable length of the milling cutter
		47	-	Neck radius of the milling cutter (RN)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read and write data of current turning tool				
	951	1	-	Tool number
		2	-	Tool length XL
		3	-	Tool length YL
		4	-	Tool length ZL
		5	-	Tool length oversize DXL
		6	-	Oversize in tool length DYL
		7	-	Tool length oversize DZL
		8	-	Tooth radius (RS)
		9	-	Tool orientation (TO)
		10	-	Angle of spindle orientation (ORI)
		11	-	Tool angle P_ANGLE
		12	-	Point angle T_ANGLE
		13	-	Recessing width CUT_WIDTH
		14	-	Type (e.g. roughing, finishing, threading, recessing or button tool)
		15	-	Length of cutting edge CUT_LENGTH
		16	-	Compensation of workpiece diameter WPL-DX-DIAM in the working plane coordinate system WPL-CS
		17	-	Compensation of workpiece diameter WPL-DZL in the working plane coordinate system WPL-CS
		18	-	Recessing width oversize
		19	-	Cutting radius oversize
		20	-	Rotation around spatial angle B for offset recessing tools

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Data of the currently active dresser				
	952	1	-	Tool number
		2	-	Tool length XL
		3	-	Tool length YL
		4	-	Tool length ZL
		5	-	Oversize for tool length DXL
		6	-	Oversize for tool length DYL
		7	-	Oversize for tool length DZL
		8	-	Cutter radius
		9	-	Cutting position
		13	-	Cutter width for plate or roll
		14	-	Type (e.g. diamond, plate, spindle, roll)
		19	-	Cutter radius oversize
		20	-	Shaft speed of a dressing spindle or roll

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Freely available memory area for tool management				
	956	0-9	-	Freely available data area for tool management. The data is not reset when the program is aborted.
Tool usage and tooling				
	975	1	-	Tool usage test for the current NC program: Result –2: Test not possible, function disabled in the configuration Result –1: Test not possible, tool usage file missing Result 0: Test OK, all tools available Result 1: Test not OK
		2	Line	Check availability of the tools required in the pallet from line IDX in the current pallet table. –3 = No pallet is defined in row IDX, or function was called outside of pallet editing –2 / –1 / 0 / 1 see NR1
Lift off the tool at NC stop				
	980	3	-	(This function is obsolete—HEIDENHAIN recommends not to use it any longer. ID980 NR3 = 1 is equivalent to ID980 NR1 = –1, ID980 NR3 = 0 has the same effect as ID980 NR1 = 0. Other values are not permissible.) Enable lift-off to the value defined in CfgLiftOff: 0 = Lock lift-off function 1 = Enable lift-off function
Touch probe cycles and coordinate transformations				
	990	1	-	Approach behavior: 0 = Standard behavior 1 = Approach probing position without compensation. Effective radius, set-up clearance is zero
		2	16	Automatic / Manual machine operating modes
		4	-	0 = Stylus not deflected 1 = Stylus deflected
		6	-	TT tool touch probe active? 1 = Yes 0 = No
		8	-	Momentary spindle angle in [°]
		10	QS parameter no.	Determine the tool number from the tool name. The return value depends on the rules configured for the search of the replacement tool.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				If there are multiple tools with the same name, the first tool from the tool table will be selected. If the tool selected by these rules is locked, a replacement tool will be returned. –1: No tool with the specified name found in the tool table or all qualifying tools are locked.
		16	0	0 = Transfer control over the channel spindle to the PLC, 1 = Assume control over the channel spindle
			1	0 = Pass tool spindle control to the PLC, 1 = Take control of the tool spindle
		19	-	Suppress touch prove movement in cycles: 0 = Movement will be suppressed (CfgMachineSimul/simMode parameter not equal to FullOperation or **Test Run** operating mode is active) 1 = Movement will be performed (CfgMachineSimul/simMode parameter = FullOperation, can be programmed for testing purposes)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Status of execution				
	992	10	-	Block scan active 1 = yes, 0 = no
		11	-	Block scan—information on block scan: 0 = NC program started without block scan 1 = Iniprog system cycle is run before block scan 2 = Block scan is running 3 = Functions are being updated –1 = Iniprog cycle was canceled before block scan –2 = Cancellation during block scan –3 = Cancellation of the block scan after the search phase, before or during the update of functions –99 = Implicit cancellation
		12	-	Type of canceling for interrogation within the OEM_CANCEL macro: 0 = No cancellation 1 = Cancellation due to error or emergency stop 2 = Explicit cancellation with internal stop after stop in the middle of the block 3 = Explicit cancellation with internal stop after stop at the end of a block
		14	-	Number of the last FN14 error
		16	-	Real execution active? 1 = execution, 0 = simulation
		17	-	2-D graphics during programming active? 1 = yes 0 = no
		18	-	Live programming graphics (**AUTO DRAW** soft key) active? 1 = Yes 0 = No
		20	-	Information on combined milling/turning mode of operation: 0 = Milling (after **FUNCTION MODE MILL**) 1 = Turning (after **FUNCTION MODE TURN**) 10 = Execute the operations for the turning-to-milling transition 11 = Execute the operations for the milling-to-turning transition
		30	-	Interpolation of multiple axes permitted? 0 = No (e.g. for straight cut control) 1 = yes

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		31	-	R+/R– possible/permitted in MDI mode? 0 = No 1 = Yes
		32	0	Cycle call possible/permitted? 0 = No 1 = Yes
			Cycle number	Single cycle enabled: 0 = No 1 = Yes
		40	-	Copy tables in **Test Run** operating mode? Value 1 will be set when a program is selected and when the **RESET+START** soft key is pressed. The **iniprog.h** system cycle will then copy the tables and reset the system datum. 0 = no 1 = yes
		101	-	M101 active (visible condition)? 0 = no 1 = yes
		136	-	M136 active? 0 = no 1 = yes

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Activate machine parameter subfile				
	1020	13	QS parameter no.	Has a machine parameter subfile with path from QS number (IDX) been loaded? 1 = Yes 0 = No
Configuration settings for cycles				
	1030	1	-	Display the **Spindle does not rotate** error message? **(CfgGeoCycle/displaySpindleErr)** 0 = no, 1 = yes
		2	-	Display the **Check the algebraic sign for depth!** error message? **(CfgGeoCycle/displayDepthErr)** 0 = no, 1 = yes
Data transfer between HEIDENHAIN cycles and OEM macros				
	1031	1	0	Component monitoring: counter of the measurement. Cycle 238 Measure machine data automatically increments this counter.
			1	Component monitoring: Type of measurement –1 = No measurement. Writing of the value with FN17 concludes Cycle 238. 0 = Circular test 1 = Waterfall diagram 2 = Frequency response 3 = Envelope curve spectrum
			2	Component Monitoring: Index of the axis from **CfgAxes\MP_axisList**
			3 – 9	Component monitoring: further arguments depend on the measurement
		100	-	Component monitoring: optional names of the monitoring tasks, as specified in **System\Monitoring\CfgMonComponent**. After completion of the measurement, the monitoring tasks stated here are executed consecutively. When assigning the input parameters, remember to separate the listed monitoring tasks by commas.
User settings for the user interface				
	1070	1	-	Feed rate limit of soft key FMAX; 0 = FMAX is inactive
Bit test				
	2300	Number	Bit number	This function checks whether a bit has been set in a number. The number to be checked is transferred as NR, the bit to be searched for as IDX, with IDX0 designating the least significant bit. To call this function for large numbers, make sure to

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				transfer NR as a Q parameter. 0 = Bit not set 1 = Bit set
Read program information (system string)				
	10010	1	-	Path of the current main program or pallet program.
		2	-	Path of the NC program shown in the block display.
		3	-	Path of the cycle selected with **SEL CYCLE** or **CYCLE DEF 12 PGM CALL**, or path of the currently active cycle
		10	-	Path of the NC program selected with **SEL PGM "..."**.
Indexed access to QS parameters				
	10015	20	QS parameter no.	Reads QS(IDX)
		30	QS parameter no.	Returns the string that you obtain if you replace anything except for letters and digits in QS(IDX) by '_'.
Read channel data (system string)				
	10025	1	-	Name of machining channel (key)
Read data for SQL tables (system string)				
	10040	1	-	Symbolic name of the preset table.
		2	-	Symbolic name of the datum table.
		3	-	Symbolic name of the pallet preset table.
		10	-	Symbolic name of the tool table.
		11	-	Symbolic name of the pocket table.
		12	-	Symbolic name of the turning tool table

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed in the tool call (system string)				
	10060	1	-	Tool name
Read machine kinematics (system strings)				
	10290	10	-	Symbolic name of the machine kinematics from Channels/ChannelSettings/CfgKinList/kinCompositeModels programmed in **FUNCTION MODE MILL** or **FUNCTION MODE TURN**.
Traverse range switchover (system string)				
	10300	1	-	Key name of the last active range of traverse
Read current system time (system string)				
	10321	1 - 16	-	1: DD.MM.YYYY hh:mm:ss 2: D.MM.YYYY h:mm 3: DD.MM.YY hh:mm 4: YYYY-MM-DD hh:mm:ss 5: YYYY-MM-DD hh:mm 6: YYYY-MM-DD h:mm 7: YY-MM-DD h:mm 8: DD.MM.YYYY 9: D.MM.YYYY 10: D.MM.YY 11: YYYY-MM-DD 12: YY-MM-DD 13: hh:mm:ss 14: h:mm:ss 15: h:mm 16: DD.MM.YYYY hh:mm 20: Calender week as per ISO 8601 As an alternative, you can use **DAT** in **SYSSTR(...)** to specify a system time in seconds that is to be used for formatting.
Read data of touch probes (TS, TT) (system string)				
	10350	50	-	Type of TS probe from TYPE column of the touch probe table (**tchprobe.tp**)
		70	-	Type of TT tool touch probe from CfgTT/type.
		73	-	Key name of the active tool touch probe TT from **CfgProbes/activeTT**.
Read and write data of touch probes (TS, TT) (system string)				
	10350	74	-	Serial number of the active tool touch probe TT from **CfgProbes/activeTT**.
Read the data for pallet machining (system string)				
	10510	1	-	Pallet name
		2	-	Path of the selected pallet table.
Read version ID of the NC software (system string)				

Group name	Group number ID...	System data number NO....	Index IDX...	Description
	10630	10	-	The string corresponds to the format of the version ID shown, e.g. **340590 09** or **817601 05 SP1**.
General data of the grinding wheel				
	10780	1	-	Name of wheel
Read information on unbalance cycle (system string)				
	10855	1	-	Path of the unbalance calibration table belonging to the active kinematics
Read data of the current tool (system string)				
	10950	1	-	Current tool name
		2	-	Entry from the DOC column of the active tool
		3	-	AFC control setting
		4	-	Tool-carrier kinematics
		5	-	Entry from the DR2TABLE column – file name of the compensation value table for 3D-ToolComp
Read data from FUNCTION MODE SET (system string)				
	11031	10	-	Returns the selection of the FUNCTION MODE SET <OEM mode> macro as a string.

40.5 Keycaps for keyboard units and machine operating panels

The keycaps with IDs 12869xx-xx and 1344337-xx are suitable for use on the following keyboard units and machine operating panels:

- TE 361 (FS)

The keycaps with ID 679843-xx are suitable for use on the following keyboard units and machine operating panels:

- TE 360 (FS)

Keycaps for alphabetic keyboard

	ESC	! 1	@ 2	# 3	$ 4	% 5	^ 6	& 7	* 8
ID 1286909	-08	-09	-10	-11	-12	-13	-14	-15	-16

	(9	) 0	_ -	+ =	Q	W	E	R	T
ID 1286909	-17	-18	-19	-20	-21	-22	-23	-24	-25

	Y	U	I	O	P	{ [	}]	¦ \	A
ID 1286909	-26	-27	-28	-29	-30	-31	-32	-33	-34

	S	D	F	G	H	J	K	L	: ;
ID 1286909	-35	-36	–	-38	-39	–	-41	-42	-43
ID 1344337*)	–	–	-01*)	–	–	-02*)	–	–	–

*) With tactile mark

	" '	~ `	Z	X	C	V	B	N	M
ID 1286909	-44	-45	-46	-47	-48	-49	-50	-51	-52

	< ,	> .	? /			ALT	PRT SC	
ID 1286909	-53	-54	-55	-56	-57	-58	-59	-60
ID 679843	–	–	–	-F4	–	–	-F6	–

ID 1286911	-01	-02	-03	-04	-05

ID 1286914	-01	-03

			CTRL
ID 1286915	-01	-02	-03

ID 1286917	-01

Keycaps for operating aids

	PGM MGT		ERR	CALC	MOD	HELP
ID 1286909	-61	-62	-63	-64	-65	-66
ID 679843	–	-36	–	–	–	–

Keycaps for operating modes

ID 1286909	-67	-68	-69	-70	-71	-72	-73	-74
ID 679843	–	–	-66	–	–	–	–	–

Keycaps for NC dialog

	APPR DEP	FK	CHF	L	CR	RND	CT	CC	C
ID 1286909	-75	-76	-77	-78	-79	-80	-81	-82	-83

	TOUCH PROBE	CYCL DEF	CYCL CALL	LBL SET	LBL CALL	STOP	TOOL DEF	TOOL CALL	PGM CALL
ID 1286909	-84	-85	-86	-87	-88	-89	-90	-91	-93

	SPEC FCT
ID 1286909	-92
ID 679843	-D6

Keycaps for axis input and value input

	X	Y	Z	IV	V	P	I	A	C
	Orange	Orange	Orange	Orange	Orange	Orange	Orange	Orange	Orange
ID 1286909	-94	-95	-96	-97	-98	-99	-0A	-4K	-4L
ID 679843	-C8	-D3	-53	-32	-31	–	–	-54	-88

	7	8	9	4	5	1	2	3	0
ID 1286909	-0B	-0C	-0D	-0E	–	-0G	-0H	-2L	-2M
ID 1344337*)	–	–	–	–	-03*)	–	–	–	–

*) With tactile mark

	.	−/+		Q	CE	DEL	NO ENT	END	
ID 1286909	-0K	-0L	-0M	-2N	-0P	-2P	-0R	-0S	-3N

				ESC	INS		i		DEL
ID 1286909	-3P	-3R	-3S	-4S	-4T	-0N	-3T	-3U	-3V
ID 679843	–	–	–	–	–	-E2	–	–	–

	>>	W	B	U
		Orange	Orange	Orange
ID 1286909	-3W	–	–	–
ID 679843	–	-55	-C9	-D4

		ENT
ID 1286914	-02	-04

Keycaps for navigation

		HOME	PG UP		GOTO		END	PG DN
ID 1286909	-0T	-0U	-0V	-0W	–	-0Y	-0Z	-1A
ID 1344337*)	–	–	–	–	-04*)	–	–	–

*) With tactile mark

ID 1286909	-1B	-1C
ID 679843	-42	-41

Keycaps for machine functions

	IV+	Z+	Y+	V+	VI+	X+			Y−
ID 1286909	-1D	-1E	-1F	-1G	-1H	-1K	-1L	-1M	-1N
ID 679843	-09	-07	-05	-11	-13	-03	-16	-17	-06

	IV−	VI−			FN 1				
ID 1286909	-1P	-1R	-1S	-1T	-1U	-1V	-1W	-1X	-1Y
ID 679843	-10	-14	-23	-22	-24	-29	-02	-21	-20

	FN 2			FN 3					X−
							Red	Green	
ID 1286909	-1Z	-2A	-2B	-2C	-2D	-2E	-2H	-2K	-2R
ID 679843	-25	-28	-01	-26	-27	-30	-57	-56	-04

		Z−	V−	+	−				
ID 1286909	–	-2T	-2U	-2Z	-3A	-3E	-3F	-3G	-3H
ID 1344337*)	-05*)	–	–	–	–	–	–	–	–
ID 679843	-15	-08	-12	-59	-60	-40	-73	-76	-74

*) With tactile mark

						C+		C−	
ID 1286909	-3L	-3M	-3X	-3Y	-3Z	-4A	-4B	-4C	-4D
ID 679843	-C6	-75	-46	-47	-F2	-67	-51	-68	-99

	W+	W−		A+	A−	B+	B−		
			Red					Red	Red
ID 1286909	-4E	-4F	-4H	-4M	-4N	-4P	-4R	-4U	-06
ID 679843	-B8	-B7	-45	-69	-70	-B2	-B1	-52	-18

	Green	Red	Red						
ID 1286909	-07	-2F	-2G	-2V	-2W	-2X	-2Y	-3B	-3C
ID 679843	-19	–	–	–	–	–	–	–	–

				Y−	Y+	Z+	Z−	Y+	Y−
ID 1286909	-3D	-3K	-4G	–	–	–	–	–	–
ID 679843	–	–	–	-43	-44	-91	-92	-93	-94

	U–	U+	Y–	Y+	Z+	Z–	X–	X+	X+
ID 679843	-B3	-B4	-B5	-B6	-B9	-C1	-C2	-C3	-C4
	U–	U+	Y–	Y+	Z+	Z–	X–	X+	X+
ID 679843	-C5	-D9	-E1	-61	-62	-63	-64	-A2	-A3
	FCT A	FCT B		FCT C					
ID 679843	-95	-96	-A1	-C7	-A4	-A5	-A6	-A9	-E3
	FCT RC		1	2	%	%	100%		Green
ID 679843	-E4	-E6	-E7	-E8	-48	-49	-50	-65	-71
	I Green	NC I Green	0 Red	NC 0 Red	T				
ID 679843	-D8	-90	-89	-D7	-72	-F3	-97	-98	-E5

Other keycaps

			Orange	Green	Red				
ID 1286909	-01	-02	-05	-03	-04	–	–	–	–
ID 679843	-33	-34	-35	–	–	-38	-39	-A7	-A8

	0	ENT
ID 679843	-D5	F5

> If you need keycaps with additional symbols, please contact HEIDENHAIN.

41 NC error messages

Applies to:
81762x-16, 34059x-16, 81760x-16, 68894x-16, 54843x-16

Date:
02/2022

List of All Error Messages

Error number	Description
120-0006	**Error message** Configuration of soft key not readable **Cause of error** The specified soft key is not contained in the configuration data. **Error correction** Check the configuration data.
120-0007	**Error message** Configuration of layer not readable **Cause of error** The configured soft-key row (layer) canot be read. **Error correction** Check the configuration data.
120-0008	**Error message** Cycle or query %1 unknown **Cause of error** The specified cycle or cycle dialog was not found in the configuration data. **Error correction** Check the configuration data.
120-000A	**Error message** Cannot generate menu **Cause of error** Software problem in the user interface **Error correction** Inform your service agency.

Error number	Description
120-000B	**Error message** Q Parameter %1: Cannot read or write value **Cause of error** Software problem in the user interface **Error correction** Inform your service agency.
120-000C	**Error message** Soft-key group without first element **Cause of error** In the configuration data a soft-key group is specified, but no soft key is marked as "first." **Error correction** Edit the configuration data.
120-000D	**Error message** Soft-key type is not supported **Cause of error** An illegal soft-key type was used within a cycle dialog. **Error correction** Edit the configuraiton data.
120-000E	**Error message** Invalid resource ID **Cause of error** Software problem in the user interface **Error correction** Inform your service agency.
120-000F	**Error message** Invalid configuration data for cycle **Cause of error** Too many soft keys defined in a cycle dialog **Error correction** Inform your service agency.
120-0013	**Error message** Error in the user interface **Cause of error** Software problem in the user interface. **Error correction** Inform your service agency.

Error number	Description
120-0016	**Error message** **Cause of error** Internal software error with front end/dialogs. **Error correction** Inform your service agency.
120-001E	**Error message** Can't connect to network: %1 %2 %3 **Cause of error** Unable to connect with a network drive defined in the network management. **Error correction** - Use the program manager to open the network management (NETWORK soft key) - Press the MOD key and enter the network code number NET123. - Enter all required data for the network connection (DEFINE NETWORK CONNECTN. soft key). - Check the correctness and spelling of the input data for the network connection
120-001F	**Error message** File '%1' not found **Cause of error** A required file could not be found at the specified location. **Error correction** Check the spelling of the path and file name, for example. If possible, copy the file to the required directory.
120-002E	**Error message** Path '%1' not found **Cause of error** A required drive or directory was not found. **Error correction** Check the spelling of the path.

Error number	Description
120-0041	**Error message** No files available for online help (*.CHM) **Cause of error** No context-sensitive help can be displayed because no files are available for the *.CHM online help. You have to download the *.CHM files from the HEIDENHAIN Homepage and save them on the control in the subfolder for your language. Note the information in the User's Manual. **Error correction** - Download the help files from the HEIDENHAIN homepage (www.heidenhain.de): > www.heidenhain.de > Documentation and information > User's documentation - Unpack the ZIP file and transfer the *.CHM files to the control
125-0067	**Error message** Error when saving the screen content in the file %1 **Cause of error** There was an error when saving the screen content in the file. **Error correction** Delete all unneeded files to increase memory capacity, or inform the service agency for your control.
125-0068	**Error message** Source file %1 not found **Cause of error** A file in the list of example files cannot be addressed as source **Error correction** Inform your service agency
125-0069	**Error message** Example file %1 could not be copied. Error code %2 **Cause of error** A file in the list of example files could not be copied **Error correction** Inform your service agency
125-006A	**Error message** List for copying the example files not available %1 **Cause of error** The file containing the list of example files is not available **Error correction** Inform your service agency

Error number	Description
125-006C	**Error message** Error during software update: %1 **Cause of error** **Error correction**
125-006D	**Error message** The update rules are not fulfilled **Cause of error** One or more update rules were not complied with. **Error correction** Inform your machine tool builder
125-006E	**Error message** Update file invalid or does not exist **Cause of error** The update file was not found or it does not contain a valid update. **Error correction** Inform your machine tool builder
125-006F	**Error message** Checksum error during update **Cause of error** The checksums do not match the update files. **Error correction** Inform your machine tool builder
125-0070	**Error message** Invalid signature entry during update **Cause of error** An invalid signature entry was found in the update file. **Error correction** Inform your machine tool builder
125-0071	**Error message** Not enough memory. Update not possible. **Cause of error** There is not enough free memory on the data medium to save the update there. **Error correction** Vacate some memory space

Error number	Description
125-0072	**Error message** Backup file not found **Cause of error** During the software update, the control automatically makes a backup file for restoring the previous software level. The backup file was not found on the data medium. **Error correction** Change to the directory in which the backup file was saved.
125-0075	**Error message** ZIP file contains no control setup **Cause of error** A .zip file was selected for the software update that does not contain any setup files for the control. **Error correction** - Check the .zip file
125-00D4	**Error message** In the configuration data, no operating times are released for editing. **Cause of error** Although the code number was entered correctly, the operating times cannot be edited because all fields of the operating times locked against editing in the configuration data. **Error correction** Release the required fields in the configuration data.
125-0117	**Error message** The connection list is full **Cause of error** The maximum number of configurable connections has been reached. **Error correction** Please delete a connection before you add another one.
125-0149	**Error message** Traverse limits were not adopted. **Cause of error** The input for one or more traverse ranges could not be loaded. Possible cause: Value range for modulo axis was entered to be greater than 360° - An NC program is being run **Error correction** - Adapt the input values and load them again - Load the traverse limits again after program run

Error number	Description
125-014A	**Error message** Input not in effect **Cause of error** The input was not accepted. Possible causes: - You have entered an illegal character. The following characters are allowed: 1234567890.- - You have entered too many characters before or after the decimal separator. **Error correction** Check and correct the entered values.
125-014B	**Error message** Kinematic model was not switched over **Cause of error** The kinematic model was not selected. Possible causes: - An NC program is being run - Tilting of the working plane is active (3D ROT soft key, PLANE function, Cycle 19) - The kinematic model is faulty **Error correction** - Switch the kinematics again after program run - Deactivate the "Tilt Working Plane" function: - Use the 3D ROT soft key to set manual tilting to inactive, or - Use the PLANE function or Cycle 19 to deactivate tilting under program control - Inform your service agency
125-0163	**Error message** Unit of measure was not switched **Cause of error** The unit of measure for the position display could not be changed. Possible cause: - An NC program is being processed **Error correction** - Switch the unit of measure again after program run
126-0072	**Error message** Error during self-test **Cause of error** An invalid message was received during the self test. **Error correction** Inform your service agency

Error number	Description
126-0075	**Error message** Error during self-test **Cause of error** An error occurred during the self test. **Error correction** - Check the emergency-stop circuits -ES.A and -ES.B for correct wiring and proper function - Inform your service agency.
126-0076	**Error message** Error during self test **Cause of error** An error occurred during the self test. **Error correction** - Check the emergency-stop circuits -ES.A and -ES.B for correct wiring and proper function - Inform your service agency
126-007F	**Error message** Self-test cannot be started **Cause of error** Before the self-test (emergency stop test), an error occurred that prevents the test from starting: - Emergency stop error has occurred - Emergency stop button was pressed - Emergency-stop circuit is defective - Internal temperature of an HSCI component is too high - Internal temperature of an HSCI component is too low - The fan of an HSCI component is defective - The power supply to the component is missing, too low or too high - HSCI cabling is missing or defective **Error correction** - Note further messages. - Use the HSCI bus diagnostics to find out which components are reporting an error - Check the emergency-stop circuit - Check the power supply to the affected HSCI components - Check the HSCI cabling - If necessary, exchange the HSCI component - Generate the service files and inform your service agency

Error number	Description
126-010F	**Error message** The CC only runs with 500 MHz **Cause of error** IThere is at least one CC controller unit in the system that is clocked only with 500 MHz. This results in performance problems in combination with the use of adaptive control functions. **Error correction** - Check the CCs and exchange them if required. - Contact your machine tool builder
126-0110	**Error message** NC software not released for FS applications **Cause of error** - The control system (hardware) was detected as a system with functional safety (FS) from HEIDENHAIN. However, the installed NC software has not been approved for applications that use integrated functional safety (FS) from HEIDENHAIN. - The installed software is a test software or an NC software that has not been released for the application of integrated functional safety (FS) **Error correction** - Install an NC software version of your control that has been approved for applications with integrated functional safety (FS) from HEIDENHAIN. - Inform your service agency.
126-0111	**Error message** Too many switch-off ports defined **Cause of error** In the IOC file, more than one output was defined for switching the machine off after shutdown. This is not allowed. No more that one output can be defined. **Error correction** - Check the IO configuration and correct it if required
126-0112	**Error message** Switch-off port on incorrect bus system **Cause of error** The output for switching off the machine after the control shutdown was defined for the wrong bus system. The permitted bus systems are external PL assemblies with HSCI interface or internal PL assemblies. **Error correction** - Check the IO configuration and correct it if required

Error number	Description
126-0113	**Error message** PLC output for shutdown has multiple definitions **Cause of error** A PLC output for shutting down the control (switch-off port) is defined both in the IOC file as well as in the configuration data (machine parameters). Note that the entry in the configuration data has priority. **Error correction** - Check the IO configuration. - Check the configuration datum DisplaySettings->CfgShut-down (Maschinen-Parameternummer 101600) - Inform your service agency
126-0114	**Error message** Automatic switch-off not possible **Cause of error** Machine switch-off after control shutdown cannot be activated because the IO configuration does not match the actual hardware configuration. - The control will be operated in simulation mode - The IOC file does not match the hardware configuration - The options in the configuration are incorrectly set. **Error correction** - Check the hardware configuration - Check the IO configuration - Check the options - Inform your service agency
126-0115	**Error message** PLC output for switch-off is improperly defined **Cause of error** The output for switching off the machine after the control has been shutdown was incorrectly configured. - An output on the internal PL was addressed (e.g. with UEC, UMC), although no internal PL exists. - The address of the output is outside of the permissible range O0 to O30. **Error correction** Check the configuration data (machine parameter number 101600).

Error number	Description
126-0116	**Error message** PLC output for switch-off is improperly configured **Cause of error** The PLC output for switching off the machine after the control has been shutdown was incorrectly configured. - An output on an HSCI device was address, although not device is connected to the control over HSCI. - The addressed HSCI device is not a PL. - The addressed output does not exist on the PL. **Error correction** - Check the configuration data (machine parameter number 101600) - Inform your service agency
126-0117	**Error message** PLC output for switch-off is improperly configured **Cause of error** The PLC output for switching off the machine after the control has been shutdown was incorrectly configured. The settings in the configuration datum CfgShutdown are not applicable for an integrated PL (e.g. with UEC, UMC) nor for a PL connected over HSCI (e.g. PLB 62xx). Settings for output on internal PL: - powerOffDevice: Unassigned - powerOffSlot: Unassigned - powerOffPort: Number of the output on internal PL (value range: 0 to 30) Settings for output on a PL connected over HSCI: - powerOffDevice: Bus address of the PL - powerOffSlot: Slot number of the module (0 for system module and UEC11x) - powerOffPort: Number of the output to be switched **Error correction** - Check the configuration data (machine parameter number 101600) - Inform your service agency
126-0118	**Error message** PLC output for switch-off is improperly configured **Cause of error** Automatic machine switch-off after control shutdown has been selected. However, no PlC output was defined for it in the configuration file nor in the IOC file. **Error correction** - Define a PLC output for switch-off in the IOC file or in the configuration file - Inform your service agency

Error number	Description
126-0119	**Error message** Key name %1 for OEM script is too long **Cause of error** - The key name used for Python script to be started during start-up is too long. Maximum of 10 characters allowed. **Error correction** - Change the configuration datum
126-011A	**Error message** Hardware combination not permitted **Cause of error** The detected hardware configuration includes an impermissible combination of CC or PL components. Examples: - CC 6106 and CC 306 - CC 306 and UEC 112 - CC 306 and older PLB, MB, or TE (not "Gen 3 ready" or not "Gen 3 exclusive") **Error correction** - Reconfigure the hardware - Inform your machine tool builder
126-011B	**Error message** Self-test error was not detected on %2 with HSCI address %3 No reaction during self-test on %2 with HSCI address %3 Self-test was not executed on %2 with HSCI address %3 **Cause of error** During the device self-test, an expected message was not received. **Error correction** - Check all cable connections from and to this device - Check the device and, if necessary, exchange it - Inform your service agency
126-011E	**Error message** Missing authorization **Cause of error** You do not have the right to execute the Commissioning mode (current controller adjustment / capturing the field angle). **Error correction** - Acquire the NC.SetupDrive right, for example by entering the password of an authorized user - Then start the function again

Error number	Description
126-011F	**Error message** Safety not ensured **Cause of error** Configuration data relevant to safety functions of the system were changed and have not been accepted yet. **Error correction** Please pay attention to the following information about parameter sets in which configuration data was changed. Perform an acceptance test of the functional safety for this system according to the guidelines of the OEM when the respective parameter sets are active. Shut the system down; while doing so, confirm that the functional safety was successfully tested and accepted.
126-0120	**Error message** Safe configuration data %1 changed **Cause of error** Configuration data relevant to safety functions of the system were changed in the indicated parameter set and have not been accepted yet. **Error correction** Perform an acceptance test of the functional safety for this system according to the guidelines of the OEM while the indicated parameter set is active. Shut the system down; while doing so, confirm that the functional safety was successfully tested and accepted.
126-0129	**Error message** Automatic switchover in simulation mode DriveSimul **Cause of error** The control was automatically switched to the "DriveSimul" simulation mode. Possible causes: - There is no controller unit (CC) in the system. - Components of drive generation 3 with external safety are being used but without a PAE module. **Error correction** - Check the hardware components installed. - Restart the control. - Set the CfgMachineSimul/MP_simMode setting to "DriveSimul". - Inform your service agency.

Error number	Description
126-012A	**Error message** Automatic switchover in simulation mode DriveAndEmS-topSimul **Cause of error** The control was automatically switched to the "DriveAn-dEmStopSimul" simulation mode. Possible cause: - No system PL was detected **Error correction** - Check the hardware components installed - Restart the control. - Set the CfgMachineSimul/MP_simMode setting to "Drive-AndEmStopSimul". - Inform your service agency.
126-012B	**Error message** Automatic switchover in simulation mode FullSimul **Cause of error** The control was automatically switched to the "FullSimul" simulation mode. Possible causes: - Neither a PL nor a machine operating panel was detected in the system. - No devices were detected at the HSCI bus. - The CfgMachineSimul/MP_simMode setting does not match the real hardware components installed. **Error correction** - Check the hardware components installed, especially the HSCI cabling and the supply voltages of the HSCI compo-nents. - Restart the control. - Set the CfgMachineSimul/MP_simMode setting to "FullSimul". - Inform your service agency.
126-012E	**Error message** Too many UM units connected to one CC: CC index %1 **Cause of error** - Too many UM inverters are connected to the stated CC controller unit. - For each CC, the maximum number of UM inverters (or motor connections) is limited to the number of axes possible on the CC. **Error correction** - Distribute the UM inverters over other CC controller units or adapt the configuration - Remove UM inverters that aren't being used (or use one-axis modules instead of two-axis modules) - Inform your service agency

Error number	Description
126-012F	**Error message** Component: %1 with serial number: %2 not planned in project **Cause of error** In a system with the Gen 3 generation of drives, all components connected to the HSCI bus and all power modules must be included in the IOconfig project. **Error correction** Add the missing component to your IOconfig project.
126-0130	**Error message** Project planning with IOconfig incomplete **Cause of error** In a system with the Gen 3 drives, all components connected to the HSCI bus and all power modules must be included in the IOconfig project. **Error correction** Add the missing components in your IOconfig project, and restart the control.
126-0131	**Error message** TNCdiag cannot be started **Cause of error** Internal error **Error correction** Inform your service agency
126-0132	**Error message** Too many HSCI devices connected. Current: %1; permitted: %2 **Cause of error** Too many devices are connected to the HSCI bus. **Error correction** - Modify the machine configuration in order to avoid exceeding the maximum number of HSCI participants - Inform your machine manufacturer
126-0133	**Error message** Too many CC/UxC devices connected. Current: %1; permitted: %2 **Cause of error** Too many CC, UEC, or UMC units are connected to the HSCI bus. **Error correction** - Modify the machine configuration in order to avoid exceeding the maximum number of HSCI participants - Inform your machine manufacturer

Error number	Description
126-0134	**Error message** Too many PLB/MB/TE/UxC devices connected. Current: %1; perm.: %2 **Cause of error** Too many PLB, MB, TE, UEC, or UMC units are connected to the HSCI bus. **Error correction** - Modify the machine configuration in order to avoid exceeding the maximum number of HSCI participants - Inform your machine manufacturer
126-0135	**Error message** Too many MB/TE devices connected. Current: %1; permitted: %2 **Cause of error** Too many MB or TE operating panels are connected to the HSCI bus. **Error correction** - Modify the machine configuration in order to avoid exceeding the maximum number of HSCI participants - Inform your machine manufacturer
126-0136	**Error message** Too many UVR devices connected. Current: %1; permitted: %2 **Cause of error** Too many UVR devices are connected to the HSCI bus. **Error correction** - Modify the machine configuration in order to avoid exceeding the maximum number of HSCI participants - Inform your machine manufacturer
126-0137	**Error message** Too many I/O terminals present. Current: %1; permitted: %2 **Cause of error** There are too many I/O terminals present on the PLB, MB, TE, UEC, or UMC units. Functionally safe terminals count as one terminal. Perhaps more terminals were detected than are actually present on the units. Some units contain internal terminals that are included in this count for technical reasons. **Error correction** - Modify the machine configuration in order to avoid exceeding the maximum number of terminals - Inform your machine manufacturer

Error number	Description
130-0001	**Error message** Processor check error **Cause of error** System error **Error correction** Inform your service agency.
130-0002	**Error message** **Cause of error** System error **Error correction** Inform your service agency.
130-0066	**Error message** **Cause of error** Arithmetical error. Value too small. **Error correction** Inform your service agency.
130-0067	**Error message** **Cause of error** Arithmetical error. Value too large. **Error correction** Inform your service agency.
130-0068	**Error message** **Cause of error** System error **Error correction** Inform your service agency.
130-0069	**Error message** **Cause of error** System error **Error correction** Inform your service agency.
130-006A	**Error message** **Cause of error** System error **Error correction** Inform your service agency.

Error number	Description
130-006B	**Error message** **Cause of error** System error **Error correction** Inform your service agency.
130-006C	**Error message** **Cause of error** System error **Error correction** Inform your service agency.
130-006D	**Error message** **Cause of error** System error **Error correction** Inform your service agency.
130-006F	**Error message** **Cause of error** System error **Error correction** Inform your service agency.
130-0070	**Error message** **Cause of error** System error **Error correction** Inform your service agency.
130-0096	**Error message** **Cause of error** System error with cancellation of machining **Error correction** Inform your service agency.
130-0097	**Error message** **Cause of error** System error **Error correction** Inform your service agency.

Error number	Description
130-0098	**Error message** **Cause of error** System error **Error correction** Inform your service agency.
130-0099	**Error message** File access was successful **Cause of error** Information for the user that file access was successful. **Error correction**
130-009A	**Error message** File system error **Cause of error** 1.) The path name contains more than six subdirectories. 2.) The path name contains a directory or file name with more than 16 characters. 3.) The path name contains two or more file name extensions or a file extension with more that 3 characters. 4.) A system error occurred during file access. **Error correction** 1.) and 2.) Save the file in another directory or under another, shorter name. 3.) Save the file with only one file name extension with no more than 3 characters. 4.) Inform your service agency.
130-009B	**Error message** File not found File not found: %1 **Cause of error** The named path has no file. **Error correction** Correct the given path name.
130-009C	**Error message** Illegal file name **Cause of error** An invalid path name was given (e.g. a path name containing illegal characters). **Error correction** Correct the given path name.

Error number	Description
130-009D	**Error message** Too many files open **Cause of error** The file couldn't be opened because too many files are already opened. There is a limit to the number of simultaneously open files. **Error correction** Close the files that you no longer need.
130-009E	**Error message** File access not possible **Cause of error** 1.) Access to the file was denied. 2.) The file is already being written to by another application. **Error correction** 1.) Check the access rights to the file and remove any existing write protection. 2.) Close the file in the application that has blocked access to the file.
130-009F	**Error message** File access not possible **Cause of error** The file was opened only for reading and cannot be written to. **Error correction** Save the files under another name.
130-00A0	**Error message** Directory could not be erased **Cause of error** The current directory cannot be deleted. **Error correction** Please select another directory first.
130-00A1	**Error message** Further file entry impossible **Cause of error** The TNC cannot save any more files. **Error correction** Delete any files that you no longer need.

Error number	Description
130-00A2	**Error message** File access not possible **Cause of error** The drive doesn't allow positioning in the file, or the positioning is not at the begining of a line. **Error correction** Inform your service agency.
130-00A3	**Error message** Drive not ready **Cause of error** Hardware error during file access. **Error correction** Check whether the device, e.g. a network adapter, is connected correctly.
130-00A4	**Error message** File access not possible **Cause of error** The file is already in use by another application. **Error correction** Close the file in the application that has access to the file.
130-00A5	**Error message** File access not possible **Cause of error** Access to a certain area of the file was blocked by another application. **Error correction** Close the file in the application that has blocked access to the file.
130-00A6	**Error message** Further file entry impossible **Cause of error** The file could not be saved because the data medium is full. **Error correction** Delete unneeded files on the data medium.
130-00A7	**Error message** Program incomplete **Cause of error** Unexpected end of file found. **Error correction** Ensure that the file is complete.

Error number	Description
130-00A8	**Error message** Directory access not possible **Cause of error** Access to the directory was denied. **Error correction** Check the access privileges to the file and, if required, cancel an existing write protection.
130-012C	**Error message** **Cause of error** The system is no longer consistent. **Error correction** Inform your service agency.
130-012D	**Error message** **Cause of error** The system cannot be started, since not all objects are available. **Error correction** Inform your service agency.
130-012E	**Error message** **Cause of error** The system cannot be started, since not all objects are available. **Error correction** Inform your service agency.
130-012F	**Error message** **Cause of error** The system cannot be started, since not all objects are available. **Error correction** Inform your service agency.
130-0130	**Error message** **Cause of error** Error in communication within the system **Error correction** Inform your service agency.

Error number	Description
130-0131	**Error message** **Cause of error** The system cannot be started, since not all objects are available. **Error correction** Inform your service agency.
130-0132	**Error message** **Cause of error** The system cannot be started, since not all objects are available. **Error correction** Inform your service agency.
130-0133	**Error message** **Cause of error** The system cannot be started, since not all objects are available. **Error correction** Inform your service agency.
130-0134	**Error message** **Cause of error** The system cannot be started, since not all objects are available. **Error correction** Inform your service agency.
130-0135	**Error message** **Cause of error** The system cannot be started because not all objects are available. **Error correction** Inform your service agency.

Error number	Description
130-0137	**Error message** Error: Unexpected end of process %1 **Cause of error** A started process ended irregularly. Possible causes: - Faulty script or error in implementation - Memory assigned for script is exhausted - Other system resources are exhausted **Error correction** - Check the log files of the script for clues. If required, edit the script. - You might find further information in the log files of the control and the operating system. - Increase the memory assigned for the script.
130-0190	**Error message** ClientQueue (%1) could not be opened **Cause of error** **Error correction**
130-0191	**Error message** Log file could not be saved. **Cause of error** The log file could not be saved under the specified path/file name. **Error correction** Enter another path/file name for saving.
130-0192	**Error message** ASSERTION: Consistency condition not fulfilled **Cause of error** System error **Error correction** Inform your service agency.
130-0193	**Error message** INFO: %1 **Cause of error** **Error correction**
130-0194	**Error message** **Cause of error** Error in communication within the system **Error correction** Inform your service agency.

Error number	Description
130-0195	**Error message** **Cause of error** No further time job could be started. **Error correction** Inform your service agency.
130-0196	**Error message** **Cause of error** Not enough memory available. **Error correction** Inform your service agency.
130-0197	**Error message** Invalid event class in the following error! **Cause of error** System error **Error correction** Inform your service agency.
130-0199	**Error message** OEM error without additional information **Cause of error** Error text was not found. **Error correction** Enter the error text in the error-text file and/or save the error text file in the corresponding directory.
130-019A	**Error message** Cycle error without additional information **Cause of error** Error text was not found. **Error correction** Enter the error text in the error-text file and/or save the error-text file in the corresponding directory.
130-019B	**Error message** %1 **Cause of error** **Error correction**

Error number	Description
130-019C	**Error message** Saving service files... **Cause of error** Service files are being saved for diagnostic purposes. **Error correction** Inform your service agency.
130-019D	**Error message** The error log could not be opened. NOTE: No error messages or info messages will be logged. **Cause of error** The log is write-protected. **Error correction** Remove the write protection, or rename or delete the log.
130-01A1	**Error message** Error while saving the service file **Cause of error** An error occurred while saving the service files. **Error correction** Generate the service files again. If required, use the SAVE SERVICE FILES soft key.
130-01FA	**Error message** Incorrect condition in switch statement **Cause of error** **Error correction**
130-01FB	**Error message** No acknowledgment from application %1 **Cause of error** The application does not confirm closing the trace files for diagnostic purposes. **Error correction** No error correction possible.
130-01FC	**Error message** System error System error **Cause of error** An as yet unimplemented function of a server was called. **Error correction** Inform your service agency

Error number	Description
130-01FD	**Error message** System error System error **Cause of error** A server cannot find the sender of a message. **Error correction** Inform your service agency
130-01FE	**Error message** System error System error **Cause of error** A server cannot reach the sender of a message. **Error correction** Inform your service agency
130-01FF	**Error message** System error System error **Cause of error** A software error has occurred. **Error correction** Inform your service agency
130-0201	**Error message** Configuration datum %1 - %2 contains errors **Cause of error** The given configuration datum has errors. The erroneous values were replaced by default values. **Error correction** Correct the given configuration datum or inform your machine manufacturer.
130-0202	**Error message** NC program cancelled **Cause of error** The NC program was cancelled because of an error. **Error correction** Note further error messages. If no further error messages appear, inform your service agency.

Error number	Description
130-03EE	**Error message** File exists already **Cause of error** The file cannot be generated because a file with the same name already exists. **Error correction** Save the file under another name.
130-03EF	**Error message** File access not possible **Cause of error** The file contains data in an illegible format. **Error correction** Select another file or inform your service agency.
130-03F0	**Error message** Directory not found **Cause of error** The specified directory does not exist or was deleted. **Error correction** Select another directory.
130-03F1	**Error message** Drive not found **Cause of error** The specified drive is not connected. **Error correction** Select another drive.
130-0414	**Error message** Service files could not be saved **Cause of error** An error occurred while saving the service files. **Error correction** Try to save again. If required, restart the control beforehand. Notify your service agency if the problem recurs.
130-0415	**Error message** There is no online help for the error! **Cause of error** **Error correction**

Error number	Description
130-07D1	**Error message** Error when opening/closing a zip file (%1) **Cause of error** The TNC was not able to create or close the zip file. The file might be corrupt. **Error correction** Try again to create the zip file.
130-07D2	**Error message** There was an error when saving a (zip) file **Cause of error** When creating the service files, the TNC was unable to save at least one file. **Error correction** No remedy possible.
130-07D4	**Error message** Key might be jammed **Cause of error** On or more keys were pressed for more than 5 seconds. **Error correction** If the problem continues, inform your service agency. Press the keys SHIFT, CTRL and ALT.
130-07D5	**Error message** Parameter transfer not possible **Cause of error** Could not write machine parameters because the background activity blocked access to the machine configuration. **Error correction** - Conclude the background activities. - Ensure during offset adjustment that the axis is in servo control and that the control window is reached.
130-07D6	**Error message** No response from drive "%1"! **Cause of error** The network drive is no longer ready or no longer reacts. **Error correction** - Check the network - Check whether the connected computer is active - Check the network cables and connectors - Check the activity of the Ethernet data interface. LEDs should light up or blink. - Have a network specialist check the network settings.

Error number	Description
130-07D7	**Error message** Program start or program selection has failed **Cause of error** The action was prevented by a simultaneous reconfiguration or a configuration error. **Error correction** - End the current simulation in the Test Run operating mode - Correct the configuration error
130-07D8	**Error message** The application cannot be started **Cause of error** The authorization necessary in order to run the application is missing. **Error correction**
140-0001	**Error message** Backup copy of the configuration data cannot be created. **Cause of error** When configuration data are written, a backup copy of the parameter file is created. This file could not be created. – File is write-protected - Drive is full - Problem with the hard disk **Error correction** - Remove the write protection - Delete unneeded files on the hard disk or memory card to make space.
140-0002	**Error message** Configuration file '%1' cannot be written. **Cause of error** The file cannot be opened for writing. - File is write-protected - Drive is full - Problem with the hard disk **Error correction** - Remove the write protection. - Delete unneeded files on the hard disk to make space.

Error number	Description
140-0004	**Error message** Configuration file '%1' not found. **Cause of error** A parameter file specified in the "configfiles.cfg" file was not found. - Incorrect parameter file entered - Incorrect directory for the parameter file entered - Parameter file deleted **Error correction** - Correct the "configfiles.cfg" or "configfile.cfg" file. - Specify the correct file in the update rule. - Create a parameter file or copy it into the desired directory.
140-0005	**Error message** Data object '%1' '%2' not complete **Cause of error** A configuration object in a configuration file is incomplete. One or more configuration parameters are missing. The incomplete configuration object is designated by object name and object key. Possible causes: - The existing configuration data belong to an older software version - The data file was edited manually **Error correction** - Define the update rule or inform your service agency - Use the configuration editor to completely fill out the parameter object
140-0006	**Error message** Key of the data object '%1' not initialized. **Cause of error** The key of a parameter object is missing. **Error correction** Enter the key for all parameter objects.

Error number	Description
140-000A	**Error message** Cycle ini file '%1' not found. **Cause of error** The cycle file specified in the parameter object "CfgJhPath" or "CfgOemPath" was not found. The name of the unfound file is given. - Incorrect file entered - Incorrect object entered - File deleted If the file is present, the problem could be caused by incorrect entries. - Incorrect or incomplete entries - Incorrect cycle, parameter, or text definitions **Error correction** - Correct the file information. - Create a file or copy one into the corresponding directory. - Use CycleDesign to correct the cycle definitions.
140-000B	**Error message** Unknown code word '%1' in cycle ini file. **Cause of error** An unknown keyword was found in a cycle file. - Incorrect cycle file - Incorrect version of CycleDesign - Incomplete or incorrect cycle definition **Error correction** - Use the correct cycle file. - Use the correct version of CycleDesign. - Use CycleDesign to correct the cycle definition.
140-000D	**Error message** No text found for text name '%1' **Cause of error** In a parameter file, the text name entered for a language-dependent text was not found. - Text is not yet defined - Text file is not compiled - Text file was not copied onto the target system - Incorrect name entered **Error correction** If no text has been entered: - Define the text in a resource file - Compile the text file - Copy the text file onto the target system Otherwise, enter an existing text name or correct the text name in question.

Error number	Description
140-000F	**Error message** Module '%1' not open. **Cause of error** A module is not logged on with the configuration server. - This task requires that you be logged on. - You attempted to log off. **Error correction** Inform your service agency.
140-0010	**Error message** Module '%1' has no general write access. **Cause of error** An attempt was made to set the general write protection on the configuration server. However, the write protection is already active and cannot be set again. **Error correction** Inform your service agency.
140-0011	**Error message** '%1' has no general write access. **Cause of error** A module is trying to cancel the general write protection, which however was not reserved by this module. **Error correction** Inform your service agency.
140-0012	**Error message** Unknown message '%1' for configuration server. **Cause of error** Internal software error **Error correction** Inform your service agency.
140-0013	**Error message** Write access for '%1' '%2' not reserved. **Cause of error** Internal software error **Error correction** Inform your service agency.
140-0014	**Error message** Incorrect file type '%1' with '%2'. **Cause of error** Internal software error **Error correction** Inform your service agency.

Error number	Description
140-0015	**Error message** Character string '%1' not found in '%2'. **Cause of error** An incorrect character string was found in a cycle file. - The file is defective - Incorrect version of CycleDesign in use **Error correction** - Use CycleDesign to create a new cycle file. - Use the correct version of CycleDesign.
140-0016	**Error message** ZERO message '%1' **Cause of error** - Insufficient main memory - Internal control error **Error correction** Inform your service agency.
140-0017	**Error message** Machine parameters were changed. Shut down the control and restart. **Cause of error** Configuration files have been changed that require a control reset for the changed data to take effect. **Error correction** - Stop NC programs and all traverse. - Press the emergency stop button. - Restart the control.
140-0018	**Error message** Data object '%1' '%2' already exists in file **Cause of error** A configuration data object with the specified key already exists. **Error correction** - Check whether the correct configuration data files are being used, e.g. whether one of the files is being used twice. - Check whether a data object is already available in another file. - Change the key of one of the redundant configuration data objects. - Delete one of the redundant objects.

Error number	Description
140-0019	**Error message** Internal software error **Cause of error** Internal software error. **Error correction** Inform your service agency.
140-001A	**Error message** Configuration object '%1' / '%2' not found **Cause of error** A configuration data object that is necessary for the control start does not exist. **Error correction** – Check the configuration in use – Check the configuration data
140-001B	**Error message** No change acknowledgment obtained from '%1' **Cause of error** Internal control error. Configuration data were changed. After notification of the change not all returned an acknowledgment within the required time. The module that did not acknowledge will be listed in the additional text. **Error correction** Inform your service agency.
140-001D	**Error message** Parameter change during program run or macro run **Cause of error** You attempted to change configuration data during a program run. These data cannot be changed during a program run (applies to reset, run or ref. errors). The additional information shows the name of the data object that cannot be changed during program run. **Error correction** Select NC stop for the running programs in Program Run Single Block, Test Run and programming graphics, and then an internal stop. Then save the data again.

Error number	Description
140-001E	**Error message** Parameter locked from editing **Cause of error** There are two possible causes: 1. An attempt was made to change configuration data while a notification of change is still pending. The last change and notification is still in progress. 2. An attempt was made to start a program while a notification of configuration change is still in progress. **Error correction** 1. Try again to save. 2. Restart the program.
140-001F	**Error message** Program start: Data objects with write protection not allowed **Cause of error** An attempt was made to start a program although data objectsare still write-protected or writing access was reserved for data objects. At program start, writing access must be allowed for all objects. **Error correction** Restart the program.
140-0020	**Error message** Module name missing or unknown '%1' **Cause of error** Internal software error. During a request a module did not enter its identification or entered it incorrectly. **Error correction** Inform your service agency.
140-0021	**Error message** Data object '%1' renamed to '%2' key '%3' **Cause of error** The read configuration data belong to an older level of the control. The specified data object was renamed by the control due to a rule. **Error correction** Check the values of the specified data object. If the data object is incorrect, correct it. Then save the data.

Error number	Description
140-0022	**Error message** Data object '%1' '%2' removed **Cause of error** The read configuration data belong to an older level of the control. The data object is no longer supported and was removed by the control due to a rule. **Error correction** Save the changed data.
140-0023	**Error message** Data type changed with data object '%1' '%2' Attribute '%3' **Cause of error** The read configuration data belong to an older version of control. The data type of a data object attribute was changed by the control due to a rule. **Error correction** Check the value of the specified attribute in the data object. If the value is incorrect, correct it. Then save the data.
140-0024	**Error message** Attribute renamed in data object '%1' '%2' Attribute '%3' **Cause of error** The read configuration data belong to an older level of the control. In the specified data object, the control renamed an attribute due to a rule. **Error correction** Check the value of the specified attribute in the data object. If the value is not correct, correct it. Then save the data.
140-0025	**Error message** Attribute in data object '%1' '%2' removed Attribute '%3' **Cause of error** The read configuration data belong to an older level of the control. In the specified data object, the control removed an attribute due to a rule. **Error correction** Check the values of the specified data object. If the data object is incorrect, correct it. Then save the data.

Error number	Description
140-0026	**Error message** Attribute inserted in data object '%1' '%2' Attribute '%3' **Cause of error** The read configuration data belong to an older level of the control. The specified data object is incomplete. Due to a rule, the control control inserted the missing attribute(s). **Error correction** Check the values of the specified data object. If the data object is incorrect, correct it. Then save the data.
140-0027	**Error message** Data object '%1' '%2' removed **Cause of error** The read configuration data belong to an older level of the control. The data object is still supported by the control, but was removed due to a rule. **Error correction** Save the changed data.
140-0028	**Error message** Data object '%1' '%2' inserted **Cause of error** The read configuration data belong to an older level of the control. The data object is missing in the given configuration data and was inserted due to a rule. **Error correction** Check the values of the specified data object. If the data object is incorrect, correct it. Then save the data.
140-0029	**Error message** Incorrect insertion value in data object '%1' '%2' Attribute '%3' **Cause of error** The read configuration data belong to an older level of the control. The specified data object is incomplete. Due to a rule, the control inserted the missing attribute. The value to be inserted as specified in the rule is incorrect. **Error correction** - The attribute is initialized with a default value. Check the value and correct it if necessary. - Correct the rule.

Error number	Description
140-002B	**Error message** Data file not saved **Cause of error** This warning is issued for information and appears if: - the configuration data have uncorrected syntax errors - or the configuration data were updated but the update was not yet saved - or default data were copied or a data backup was restored but the control was not yet restarted. In this condition, no configuration data are written to the data files. **Error correction** - Manually correct the fatal errors in the configuration editor by selecting the FIX SYNTAX ERROR soft key. - Fix the errors that are corrected by an update rule by selecting the CONFIG DATA soft key in the configuration editor and then saving. - Restart the control.
140-002C	**Error message** Unknown object name '%1' **Cause of error** An object name is unkown. The name might be incorrectly written, or it might not be supported by the version. **Error correction** - If the name comes from a configuration file, correct it there. - Otherwise, inform your service agency.
140-002D	**Error message** Maximum list size exceeded **Cause of error** The maximum list size has been reached. Too many configuration data objects have been added. **Error correction** - Delete configuration data objects - If this is not possible, inform your service agency

Error number	Description
140-002E	**Error message** SIK control identification faulty: %1 **Cause of error** The SIK (System Identification Key) is not suitable for this software. With this software, the control can be operated only as a programming station. Possible causes: - Control is an export version - Wrong SIK - Incorrect or unconfigured control type - Error while accessing the SIK. **Error correction** - Inform your service agency
140-002F	**Error message** General key expired **Cause of error** The machine tool builder can use a master keyword (general key) for putting the control into service that will unlock all options for a duration of 90 days. The general key has expired and is therefore no longer valid. Now the options will be active only with the fiting keywords. **Error correction** Order the required software option(s) from your service agency.
140-0030	**Error message** General key is active **Cause of error** The machine tool builder can use a master keyword (general key) for putting the control into service that will unlock all options for a duration of 90 days. This message is displayed after every start-up if the general key is active. After the time period is over, the options can be used only with the fitting keywords. If an NC program is running when the general key expires, an NC stop is issued! **Error correction** Contact the machine manufacturer. He can check in the SIK menu how long the general key will be valid, or deactivate the general key.
140-0031	**Error message** '%1' entered as password **Cause of error** **Error correction**

Error number	Description
140-0032	**Error message** Incorrect software version **Cause of error** The export code does not match this software version. **Error correction** You have to switch from the standard version to the export version or vice versa.
140-0033	**Error message** Job cannot run **Cause of error** The configuration server cannot run a job. For more information about the job, refer to the additional information. **Error correction** - Check other entries in the log: there might be additional entries that provide more information for troubleshooting. - Check the machine parameter System > PLC > CfgPlcOptions > noConfigDataLock: if the parameter is not set or is FALSE, then parameter changes are not permissible while executing a program. Set noConfigDataLock to TRUE in order to permit changes to parameters while executing a program. Refer to the Technical Manual for more information about noConfigDataLock. - Inform your service agency
140-0034	**Error message** Data object '%1' '%2' moved to file '%3' **Cause of error** The downloaded configuration data belong to an older NC software level. The data object was saved in an incorrect file and was moved to another file according to a rule. **Error correction** Save the modified data.
140-0035	**Error message** Data object '%1' '%2' divided into '%3' **Cause of error** The downloaded configuration data belong to an old NC software level. The data object was divided into one or more data objects. **Error correction** Save the modified data.

Error number	Description
140-0036	**Error message** Value of data object '%1' '%2' was changed. Attribute '%3'. **Cause of error** The downloaded configuration data belong to an old NC software level. The control modified one or more values of a data object on the basis of a rule. **Error correction** Check the values of the given attributes in the data object. If the value is incorrect, correct it. Then save the data.
140-0037	**Error message** Attribute of data object '%1' '%2' was moved. Attribute '%3'. **Cause of error** The downloaded configuration data belong to an older NC software level. The control moved an attribute of the given data object on the basis of a rule. **Error correction** Check the values of the given data object. If the data object is incorrect, correct it. Then save the data.
140-0038	**Error message** Data object '%1': Key was changed from '%2' to '%3' **Cause of error** The downloaded configuration data belong to an older version of the control. The objekt key of a data object was changed on the basis of a rule. **Error correction** Check the values of the given data object. If the data object is incorrect, correct the object. Then save the data.
140-0042	**Error message** Faulty cycle data **Cause of error** - Too many soft keys in one level - Error in menu tree - Other error **Error correction** - Distribute the soft keys over multiple levels - Configure the menu tree correctly - Note the additional information

Error number	Description
140-0043	**Error message** Too much data **Cause of error** You changed too many data objects or loaded an MP subfile that is too large. Not all changed data objects will become effective immediately. **Error correction** - Shut down and restart the control. - Divide the MP subfile
140-0044	**Error message** Shut down and restart the control **Cause of error** The trial license was deactivated. However, the temporarily enabled options remain active until the control is shut down. **Error correction** The control must be shut down and restarted.
140-0045	**Error message** Error while opening file '%1' **Cause of error** - File does not exist - Access to file failed - File is not a valid backup file **Error correction** - Select an existing file - The file must not be opened by another file - The selected file must first be created by the control as a backup file
140-0046	**Error message** Data change without effect **Cause of error** More than one MP subfile is loaded, or parameters were changed by a PLC module. An attempt was made to edit data, but the changes have no effect during control operation because they are covered over by MP subfiles or PLC changes. **Error correction** Check the active data in the 'effective data' data record. If they do not have the desired values: - Reload the MP subfile in which the changes were made - Unload other MP subfiles - Edit the data in the 'temporary files' data record

Error number	Description
140-0047	**Error message** Error while loading the backup **Cause of error** - Program Run is active - The files belong to an older software version - A file has a syntax error - The files from the backup were restored but could not be activated because they contain reset parameters with other values. **Error correction** - Stop the program - The data must be updated. Refer to the instructions for updating configuration data. - Use the "REMOVE SYNTAX ERROR"REMOVE SYNTAX ERROR" soft key to make a manual correction - The control must be shut down and restarted. If you have to restore the old data, activate the file %OEM%:\config_LastKnownGoodConfig_.zip
140-0048	**Error message** Error while creating the backup **Cause of error** - Not allowed in this data record - File cannot be created **Error correction** - Select the basis data record - Select another file
140-0049	**Error message** The software option for OEM cycles has not yet been enabled **Cause of error** An OEM cycle tree was configured, but the software option has not been enabled. **Error correction** Order the required software option from your service agency.
140-004A	**Error message** The software option for '%1' has not been enabled **Cause of error** The language selected for the required software option is not enabled. **Error correction** Order the required software option from your service agency.

Error number	Description
140-004B	**Error message** Error in finding the current control parameters **Cause of error** There was an error in the automatic calculation of the current controller parameters. **Error correction** The current controller parameters must be acquired manually.
140-004C	**Error message** System settings deleted **Cause of error** The system settings in non-volatile memory, e.g. which programs were last opened, were deleted. When the control is started it reinitializes these system settings. **Error correction** Shut the control down and restart it.
140-004D	**Error message** Password %1 defined twice **Cause of error** The entered coder number or password already exist. The double definition is not allowed. **Error correction** - Replace the entered password in the configuration (CfgOemPassword or CfgChangePassword). Use a password that has not yet been used.
140-004E	**Error message** Password %1 replaced by machine manufacturer **Cause of error** The machine manufacturer replaced the entered password with another one. **Error correction** Use the password defined by the machine manufacturer. Refer to the machine manual or contact your machine manufacturer to receive the password.

Error number	Description
140-004F	**Error message** Update of the configuration data **Cause of error** The active machine configuration is not compatible with the current version of the NC software. The configuration files belong to an older software version. **Error correction** - After a configuration backup is loaded you have to close the control down and restart it. - The code number prompt is shown during start-up. Enter the code number for the Machine Parameter Editing operating mode. - The control implements the configuration data according to the update rules. Check the changes in the machine configuration and save the edited configuration data. - Also also read the instructions for updating configuration data in the Technical Manual.
140-0050	**Error message** Impermissible data object for file **Cause of error** The data object was saved in a impermissible location. The data object either cannot be saved in a parameter file or it has to be located in a file of the PLC: or SYS: drive. **Error correction** Remove the data object completely from the file or move it into a file on another drive. Note also which access is defined for the data object.
140-0051	**Error message** Data object "%1" is faulty **Cause of error** A configuration object in a configuration file is faulty or incomplete. Possible causes: - The existing configuration data belong to an older software version. - The data file was edited manually. **Error correction** - Reset the config version and run the update again. - Use the configuration editor to correct or completely fill out the parameter object

Error number	Description
140-0052	**Error message** Installation of setup in PLCE drive failed **Cause of error** The installation of setup files into the encrypted PLCE drive has failed. **Error correction** Possible causes: - An incorrect password was entered for encrypting the setup file in PLCdesign. Make a new setup.zip file in PLCdesign with the correct password. - A target file name or target directory was entered in PLCdesign that contains illegal characters. Use only ASCII characters for target paths in "PLCE:". Make a new setup.zip file with PLCdesign. - The image file for "PLCE:" is too small for the update. Save the contents of "PLCE:", generate a larger image file, restore the backup and try the update again. - There is no encrypted "PLCE:" partition on the control. Make an encrypted partition and retry the update. - No password was found for the encrypted partition or the incorrect password was entered. Enter a correct password in the settings for the "PLCE:" partition (PLC Programming mode).
140-0053	**Error message** Configuration data object not found **Cause of error** A configuration data object that is necessary for a software update does not exist The update of the configuration data might therefore be incomplete. **Error correction** Check the machine configuration and correct it if required.
140-0054	**Error message** Error while loading the default data **Cause of error** An error occurred while loading the default data: - One or more files are still being accessed by the control - The default data incomplete or incorrect. **Error correction** - Check whether there are any more messages in the log file that indicate the possible cause of the error. - Restart the control and stop the start-up by immediately pressing the MOD key. Then load the default data - If this does not help, inform your service agency

Error number	Description
140-0055	**Error message** Faulty update rule **Cause of error** The update rule given in the additional data is faulty. The machine configuration cannot be updated with this rule. **Error correction** - Check all settings in the update rule. - If a file is named in the rule: Check whether this file is part of the present configuration. - Correct the update rule so that the machine configuration can be updated successfully.
140-0056	**Error message** Error while saving configuration data **Cause of error** An error occurred while saving the configuration data. **Error correction** - Please note the additional section shown in the internal information. - Inform your service agency.
140-0057	**Error message** Illegal value **Cause of error** - The test of the entered value found that it is incorrect or invalid. - Whether the value is valid can also depend on other machine parameters (plausibility). **Error correction** - Check the entered value and change it if necessary. If the parameter depends on other parameters, you must consider them as well. - Inform your service agency
140-0058	**Error message** Data object '%1' does not exist **Cause of error** - Error when writing data. - Given key name does not exist. **Error correction** - Check the key name and correct if necessary, or enter an existing key name. - First create a data object with this key name. - Inform your service agency.

Error number	Description
140-0059	**Error message** Too many MP partial files **Cause of error** The maximum number of subfiles was exceeded while MP subfiles were loading. **Error correction** - Unload all MP subfiles using the config editor - Restart the control - Inform your service agency
140-005A	**Error message** Faulty data in update rule **Cause of error** The data object given in the update rule is faulty. **Error correction** Check the spelling of the data object shown under “object” and correct it if necessary: - Check the spelling of all names. - Check the syntax of parentheses for completeness. Opening parentheses must be closed again. - Check the syntax of all machine parameters or attributes. Attributes must be separated by commas. Important: A comma must not precede a parenthesis! - Check the syntax of character strings. Character strings must be enclosed by quotation marks. - If quotation marks are used, they must be preceded by a backslash. Example: \”A name\” - If there is a backslash, it must be preceded by another. Example: \\ - Inform your service agency.
140-005B	**Error message** Updating of system data not permitted **Cause of error** - You tried to use the update rules to change or overwrite write-protected data/files or configuration data. - Update rules must not edit write-protected data or system data. **Error correction** - Remove the faulty update rule. You will find the faulty update rule in the error message’s additional information by pressing the INTERNE INFO soft key. - Inform your service agency.

Error number	Description
140-005C	**Error message** Faulty value for machine parameter **Cause of error** - Syntax error in the value for a machine parameter in the update rule. - Entered value has a spelling error or in not allowed for the parameter that is to be changed by the update rule. **Error correction** - Check the spelling of the value. - Check the type of value. A numerical value can contain only one number. An enumerated list must be given an existing enumeration name. - Inform your service agency.
140-005D	**Error message** Unknown name in update rule **Cause of error** - Unknown attribute name or machine parameter entered in the update rule. **Error correction** - Check the name for spelling errors. - Check the data object for whether the attribute or machine parameter really exists. - Inform your service agency.
140-005E	**Error message** Incorrect list index **Cause of error** - No index or excessively large index entered for the list in the update rule. **Error correction** - If no index was entered, enter one. - Check whether the entered index is larger than the maximum index of the associated list. Change the index if necessary. - Inform your service agency.

Error number	Description
140-005F	**Error message** Unknown file '%1' in update rule '%2' **Cause of error** - In the update rule, a file is given that is either nonexistent or is not listed in the list of configuration files. - You can find more information on the error message by pressing the INTERNAL INFO soft key. **Error correction** - The file does not exist or is not in the list of configuration files: - The control will automatically create a file and save the new data in the file. - If you do not want to use the new file, move the data to another one and delete the existing file from the list of configuration files. - Incorrect path or file name in the update rule: - Press the UPDATE RULES soft key, then check the update rule and correct it if necessary. - Configuration file exists, but is not in the list of configuration files: - Press the “CONFIG FILE LIST” soft key and add the file to the list. - Inform your service agency.
140-0060	**Error message** Unit of measure “%1” not defined **Cause of error** - Unknown unit of measure given in the attribute information. **Error correction** - Check the unit for mass and change it if necessary. Use only the default units of math. Only the measuring unit’s name can be entered, not the measuring unit itself. - Note that, depending on the unit of measure, the displayed value is converted to the unit of measure used by the control. - With machine parameters CfgOemInt and CfgOemPosition, no units of measure can be used that require conversion.
140-0062	**Error message** Configuration error while logging the user on or off **Cause of error** A configuration error occurred in the operating system when logging the user on or off. **Error correction** - Ensure that all required configuration data is available to the user, particularly the data on HOME: - Stop all NC programs - Log the user off and then on again. If errors are still being reported, please contact your service agency.

Error number	Description
140-0063	**Error message** Function with encrypted data not allowed **Cause of error** If encrypted configuration files are used, no configuration backup or restore can be executed. **Error correction** Start the backup or restore with PC tools, such as TNCremo.
140-0064	**Error message** Software option '%1' is disabled by the configuration **Cause of error** - The software option is locked by configuration. The software option is therefore not available, although it has been enabled in the SIK. - Observe any changed machine behavior (for example, when DCM collision monitoring is locked)! **Error correction** - Re-enable the software option in the SIK dialog to restore the original behavior. - Inform your service agency.
140-0065	**Error message** Load subfile through NC program **Cause of error** The selected MP subfile was loaded by an NC program from one of the simulation operating modes (e.g. Test Run). Such an MP subfile is indicated with a prefixed pound sign (#). This MP subfile can only be unloaded from the same simulation operating mode. **Error correction** - Switch to the simulation operating mode from which the MP subfile was loaded. - Start an NC program there that unloads the MP subfile. - Inform your service agency.
140-0066	**Error message** Configuration files adapted as the result of an update **Cause of error** For informational purposes: At least one of the files indicated under CfgConfigUpdate/baseFiles or CfgConfigUpdate/portionFiles was automatically adapted due to a software update. **Error correction** HEIDENHAIN recommends checking the changes that were performed automatically.

Error number	Description
140-0067	**Error message** Error while re-reading the configuration data **Cause of error** Errors occurred while reloading the configuration data. Refer to the additional information in the error window opened by the INTERNAL INFO soft key: HAS_FATAL: The data that was read includes syntax errors HAS_UPD: The data must be updated Please note: the previous configuration data of the control is still active. It is no longer possible to change the configuration data. **Error correction** Restart the control
140-0068	**Error message** File '%1' listed more than once **Cause of error** A file is listed more than once in CfgConfigDataFiles. Please note: The file can also be listed under SYS: in CfgJhConfigDataFiles. **Error correction** - Change the file name and entry in CfgConfigDataFiles
141-0003	**Error message** Key already exists **Cause of error** A data object with the entered key already exists. Therefore no new data object can be created with this key. **Error correction** - Delete old data object - or enter another key
141-0005	**Error message** Data was not fully saved **Cause of error** Some of the changed data could not be saved because it is locked from write access or an NC program is still running. **Error correction** - Stop NC program run - Try again to save, because the lock on write access might have been removed in the meantime.

Error number	Description
141-0006	**Error message** Value out of range %1 to %2 **Cause of error** The entered value is invalid or lies outside the permitted limit values. **Error correction** Enter another value.
141-0025	**Error message** Insufficient access rights for data change '%1' '%2' **Cause of error** You do not have the access rights to change data. Perhaps you are running a cycle or NC program from a drive for which you do not have access rights. The message shows the data object for which more access rights are required. **Error correction** - The data cannot be saved. - Another code number must be entered. Run the cycle or NC program from a drive for which you do have the necessary access rights. - For a software update the system code number is required.
141-0030	**Error message** No further password entries possible **Cause of error** An incorrect password was entered repeatedly to enable an option. After 10 incorrect entries no further entry is accepted. **Error correction** Inform your service agency.
141-0048	**Error message** Selection is not an MP subfile **Cause of error** You selected a directory instead of a file. This function supports the selection of only one file. **Error correction** - Either select an MP subfile (no directory) - Or use the soft key FILE / UPDATE DIRECTORY to update all the files in the directory

Error number	Description
141-004B	**Error message** Required value missing **Cause of error** A required value was not entered. **Error correction** - Enter a value at the indicated position
141-004C	**Error message** Invalid input range or incorrect format **Cause of error** The entered value does not fit the parameter format. **Error correction** - Enter a value from the valid value range for input in the associated parameter
141-004D	**Error message** Value must not be changed **Cause of error** You have entered attribute information that cannot be changed by the machine manufacturer. This input might have happened directly in a parameter file. **Error correction** Enter only the data that are offered in the dialog window for editing attribute information.
141-004E	**Error message** No line has been selected **Cause of error** Under "Selection," you have not chosen a field from the table. **Error correction** - Use the cursor to select a line in the table under "Selection." Note: It depends on the type of parameter whether a selection can be entered!
141-004F	**Error message** Function possible only with basic data **Cause of error** TEST EN **Error correction** TEST EN

Error number	Description
141-0050	**Error message** Software option [%1] cannot be disabled **Cause of error** You tried to lock a software option listed in the SIK dialog using the LOCK OPTION soft key. The selected option belongs to the control's factory default setting. Options that are available as standard cannot be locked. **Error correction** Select a software option available for this control for the LOCK OPTION function. You can find the list of available software options in the Technical Manual.
141-0141	**Error message** Copy default configuration to \\CONFIG ? **Cause of error** **Error correction**
141-0142	**Error message** Previous data are saved in \\CONFIG.BAK **Cause of error** **Error correction**
145-0001	**Error message** Circular arc not fully defined **Cause of error** The start angle of the circular arc is not defined. **Error correction**
145-0002	**Error message** Circular arc not fully defined **Cause of error** The stopping angle of the circular arc is not defined. **Error correction**
145-0003	**Error message** Circular arc not fully defined **Cause of error** Starting and stopping angles of the circular arc are not defined. **Error correction**

Error number	Description
145-0004	**Error message** System error in the geometry **Cause of error** The position of a circular spatial arc is not explicitly defined. **Error correction** Inform your service agency
145-0005	**Error message** System error in the geometry **Cause of error** Message must not contain a transformation matrix **Error correction** Inform your service agency
145-0006	**Error message** Invalid scaling factor **Cause of error** You defined different axis-specific scaling factors for the same circle. For circles, use identical axis-specific scaling factors. **Error correction**
145-0008	**Error message** Incorrect tangent **Cause of error** The specified tangent end point lies too closely to the arc. **Error correction** Correct the end point of the tangent.
145-0009	**Error message** Incorrect tangent **Cause of error** The specified tangent point lies on the circular arc. **Error correction** Correct the end point of the tangent.
145-000A	**Error message** Incorrect tangent **Cause of error** The specified tangent point lies inside the circular arc. **Error correction** Correct the end point of the tangent.

Error number	Description
145-000C	**Error message** Incorrect point **Cause of error** The given point does not line on the circular arc. **Error correction** Correct the coordinates of the point.
145-000D	**Error message** Function not yet implemented **Cause of error** Function not yet implemented. **Error correction**
145-000E	**Error message** Incorrect circular arc **Cause of error** Start point and end point of the circular arc are at different distances from the center. **Error correction** Correct the coordinates of the end point and center.
145-000F	**Error message** Incorrect circular arc **Cause of error** The distance between the start point and end point of the circular arc is too small. **Error correction** Correct the point.
145-0010	**Error message** Incorrect circular arc **Cause of error** The specified radius is too small to connect the start point and end point. **Error correction** Correct the coordinates.
145-0011	**Error message** Incorrect circular arc **Cause of error** The tangents at the start poine and end point of the circular arc are parallel. **Error correction** Correct the coordinates.

Error number	Description
145-0012	**Error message** No intersection **Cause of error** The specified curves have not intersection. **Error correction** Correct the coordinates.
145-0013	**Error message** Incorrect chamfer **Cause of error** Length of chamfer is undefined **Error correction** Enter the chamfer length
145-0014	**Error message** Incorrect chamfer **Cause of error** Chamfer too long **Error correction** Correct the chamfer length.
145-0015	**Error message** Incorrect chamfer **Cause of error** Chamfer is possible only between straight lines **Error correction** Delete the chamfer block
145-0016	**Error message** Error in rounding arc (RND) **Cause of error** Radius of rounding arc is undefined **Error correction** Correct the rounding arc
145-0017	**Error message** Error in rounding arc (RND) **Cause of error** Rounding arc radius too large **Error correction** Correct the rounding arc radius

Error number	Description
145-0018	**Error message** Incorrect approach movement (APPR) **Cause of error** Undefined length of approach movement **Error correction**
145-0019	**Error message** Incorrect approach movement (APPR) **Cause of error** Undefined side of approach **Error correction**
145-001A	**Error message** Incorrect approach movement (APPR) **Cause of error** Undefined radius of approach **Error correction**
145-001B	**Error message** Incorrect approach movement (APPR) **Cause of error** Undefined traverse angle of approach **Error correction**
145-001C	**Error message** Incorrect approach movement (APPR) **Cause of error** Radius of the transition arc in approach too large **Error correction**
145-001D	**Error message** Incorrect departure movement (DEPT) **Cause of error** Length of departure movement is undefined **Error correction**
145-001E	**Error message** Incorrect departure movement (DEPT) **Cause of error** Departure side undefined **Error correction**

Error number	Description
145-001F	**Error message** Incorrect departure movement (DEPT) **Cause of error** Radius of departure is undefined **Error correction**
145-0020	**Error message** Incorrect departure movement (DEPT) **Cause of error** Traverse angle of departure movement is undefined **Error correction**
145-0021	**Error message** Incorrect departure movement (DEPT) **Cause of error** Radius of the transitional arc in the departure movement is too large **Error correction**
145-0022	**Error message** Incorrect departure movement (DEPT) **Cause of error** End point of departure movement is undefined **Error correction**
145-0023	**Error message** System error in the geometry **Cause of error** Access to a vector component with invalid index **Error correction** Inform your service agency.
145-0024	**Error message** System error in the geometry **Cause of error** An attempt was made to normalize a null vector. **Error correction** Inform your service agency.
145-0025	**Error message** System error in the geometry **Cause of error** Access to a matrix element with invalid index. **Error correction** Inform your service agency.

Error number	Description
145-0026	**Error message** System error in the geometry **Cause of error** Access to the column of a matrix with invalid index. **Error correction** Inform your service agency.
145-0027	**Error message** System error in the geometry **Cause of error** You attempted to invert a singular matrix. **Error correction** Inform your service agency.
145-0028	**Error message** System error in the geometry **Cause of error** No conversion possible **Error correction** Inform your service agency.
145-002F	**Error message** Translation error **Cause of error** The definition of the orientation in a coordinaten transformation is incorrect. **Error correction** Correct the definition.
145-0030	**Error message** Translation error **Cause of error** In a cordinaten transformation the Y direction cannot be calculated. **Error correction** Correct the definition.
145-0031	**Error message** Geometry error **Cause of error** The distance between two points is too small for one calculation. **Error correction**

Error number	Description
145-0032	**Error message** System error in the geometry **Cause of error** Division by zero during an internal calculation **Error correction** Inform your service agency.
145-0033	**Error message** System error in the geometry **Cause of error** An internal calculation resulted in a circle with negative radius. **Error correction** Inform your service agency.
145-0034	**Error message** System error in the geometry **Cause of error** For an ellipse or an elliptical arc, a function was called that is permitted only for circles or circular arcs. **Error correction** Inform your service agency.
145-0035	**Error message** System error in the geometry **Cause of error** Internal software error. **Error correction** Inform your service agency.
145-0036	**Error message** System error in the geometry **Cause of error** Two points are so close together in a geometric figure that the polar vector is too inexact. **Error correction** Inform your service agency.
145-0037	**Error message** Circle incorrectly programmed **Cause of error** No circle is defined through the given point. **Error correction** Correct the coordinates of the points.

Error number	Description
145-0038	**Error message** System error in the geometry **Cause of error** In an internal calculation, an ellipse has degenerated into a line segment. Some operations cannot be performed for such degenerated ellipses. **Error correction** Inform your service agency.
145-0039	**Error message** Error in rounding arc (RND) **Cause of error** An attempt was made to place a rounding radius between two directionally opposed parallel lines. **Error correction** Correct the coordinates of the lines.
145-003A	**Error message** Undefined rotational direction of contour **Cause of error** You tried to learn the rotational direction of a contour that has none defined. **Error correction** Ensure that the contour is closed and free of gaps.
160-0001	**Error message** System error in interpreter **Cause of error** Internal software error **Error correction** Inform your service agency.
160-0003	**Error message** Block format incorrect **Cause of error** Keyword or G function not programmed. **Error correction** Edit the NC program.
160-0004	**Error message** System error in interpreter **Cause of error** Internal software error **Error correction** Inform your service agency.

Error number	Description
160-0007	**Error message** System error in interpreter **Cause of error** Error in internal cycle **Error correction** Inform your service agency.
160-000B	**Error message** CYCL DEF not defined **Cause of error** You have programmed a cycle call without first defining the cycle, or you tried to call a DEF-active cycle. **Error correction** Define the cycle before calling it.
160-000D	**Error message** Error when opening the file '%1' **Cause of error** An error occurred while a file was being opened. **Error correction** Ensure that the file exists, that the given path is correct, and that the file has a readable format.
160-0018	**Error message** Jump to label 0 not permitted **Cause of error** In a LBL CALL (ISO: L 0,0) block of a part program or in a jump instruction (parametric calculation) you attempted to program a jump to the label 0. **Error correction** Edit the part program.
160-001A	**Error message** Block format incorrect **Cause of error** The program section repeat was incorrectly programmed. **Error correction** Edit the NC program.

Error number	Description
160-001B	**Error message** Label number not found **Cause of error** With LBL CALL (ISO: L x,x), you tried to call a label that does not exist. **Error correction** Change the number in the LBL CALL block or insert the missing label (LBL SET).
160-001C	**Error message** Label number already assigned **Cause of error** You attempted to program the same label number in several LBL SET (ISO: G98 Lxx) blocks in a part program. **Error correction** Edit the part program.
160-001D	**Error message** Block format incorrect **Cause of error** A positioning movement was programmed without type of interpolation. **Error correction** Edit the NC program.
160-001F	**Error message** Wrong axis programmed **Cause of error** An axis programmed in the NC block is not configured. **Error correction** Edit the NC program.
160-0020	**Error message** Wrong axis programmed **Cause of error** No nominal/actual position could be found for the axis specified in the FN18 block (ISO: D18). **Error correction** Check the index of the system data.

Error number	Description
160-0021	**Error message** Pole incorrectly defined **Cause of error** In the NC program both coordinates of a pole must be programmed in the plain. Either you forgot one coordinate, entered more than two coordinates, or programmed one coordinate twice. **Error correction** Check the pole programming in the NC program.
160-0022	**Error message** Block format incorrect **Cause of error** In a linear block, you programmed the polar radius twice **Error correction** Edit the NC program
160-0023	**Error message** Block format incorrect **Cause of error** A radius was programmed in a polar circular interpolation CP (ISO:G12/G13/G15), but the radius is defined by the distance of the starting point to the pole. **Error correction** Edit the NC program.
160-0024	**Error message** Block format incorrect **Cause of error** The radius is missing for a Circle with Radius block (CR, ISO: G02, G03). **Error correction** Edit the part program.
160-0025	**Error message** Block format incorrect **Cause of error** The arc direction of rotation was incorrectly programmed. **Error correction** Edit the NC program.

Error number	Description
160-0026	**Error message** Direction of rotation missing **Cause of error** A circle was programmed without direction of rotation. **Error correction** Always program a direction of rotation (DR).
160-0028	**Error message** Invalid Q parameter **Cause of error** The given index for a Q parameter is out of range. **Error correction** Edit the NC program.
160-0029	**Error message** Block format incorrect **Cause of error** The programmed NC syntax is not supported. **Error correction** Edit the NC program.
160-002A	**Error message** Block format incorrect **Cause of error** The programmed NC syntax is not supported. **Error correction** Edit the NC program.
160-002B	**Error message** Block format incorrect **Cause of error** The programmed NC syntax is not supported. **Error correction** Edit the NC program.
160-002C	**Error message** Block format incorrect **Cause of error** A syntactically incorrect NC block was programmed. **Error correction** Edit the NC program.

Error number	Description
160-0032	**Error message** Arithmetical error **Cause of error** Incorrect Q parameter calculation: Division by 0, square root of a negative number, or similar error **Error correction** Check the input values.
160-0036	**Error message** NC block not found **Cause of error** The block specified for mid-program startup was not found. **Error correction** Enter another target for the mid-program startup.
160-003C	**Error message** CYCL DEF incomplete **Cause of error** You programmed an incomplete cycle definition or inserted other NC blocks between cycle blocks. **Error correction** Edit the NC program.
160-003D	**Error message** Block format incorrect **Cause of error** The programmed syntax element is not allowed in this NC block. **Error correction** Edit the NC program.
160-003E	**Error message** Block format incorrect **Cause of error** The programmed syntax element is not allowed in this NC block. **Error correction** Edit the NC program.
160-0048	**Error message** Block format incorrect **Cause of error** The programmed NC syntax is not supported by this control. **Error correction** Edit the NC program.

Error number	Description
160-0049	**Error message** Touch point inaccessible **Cause of error** No signal came from the touch probe within the measuring distance. **Error correction** Pre-position as appropriate and repeat the probing process.
160-0054	**Error message** Block format incorrect **Cause of error** An incorrect system data number was entered in the function FN18 ID2000. **Error correction** Edit the NC program.
160-0055	**Error message** Tool change faulty **Cause of error** An incorrect tool was enabled. **Error correction** Inform your machine tool builder.
160-0056	**Error message** Tool axis is missing **Cause of error** You programmed a positioning block with tool radius compensation without first calling a tool. **Error correction** Edit the NC program.
160-0058	**Error message** Cycle for TOOL CALL not defined **Cause of error** The tool definition cycle was not defined. **Error correction** Inform your machine tool builder.
160-0059	**Error message** Cycle for TOOL DEF not defined **Cause of error** The tool definition cycle was not defined. **Error correction** Inform your machine tool builder.

Error number	Description
160-005A	**Error message** Unknown tool type '%1' **Cause of error** An undefined tool type was found in function FN17 ID950. **Error correction** Inform your machine tool builder.
160-005D	**Error message** Block format incorrect **Cause of error** The data type of the column given in an SQL BIND command does not agree with that of the given parameter. **Error correction** Check the table definition and edit the NC program.
160-0060	**Error message** Parameter not connected with a column **Cause of error** An attempt was made to use SQL BIND to dissolve a link that does not exist. **Error correction** Edit the NC program.
160-0061	**Error message** System error in interpreter **Cause of error** Internal software error **Error correction** Inform your service agency.
160-0063	**Error message** Block format incorrect **Cause of error** The programmed syntax element is not allowed in this NC block. **Error correction** Edit the NC program.

Error number	Description
160-0064	**Error message** Table access failed **Cause of error** - The syntax of the programmed SQL statement is incorrect - The given table could not be opened - The given table is incorrectly defined - The symbolic name of the table is not defined - The table does not contain the given columns - A data record could not be read because it is locked **Error correction** Check the table definition and edit the NC program
160-0065	**Error message** Block format incorrect **Cause of error** An unfulfillable condition was given in the FN20 function. **Error correction** Edit the NC program.
160-0067	**Error message** Faulty access to table **Cause of error** An internal control error has occurred. **Error correction** Inform your service agency.
160-0068	**Error message** Block format incorrect **Cause of error** A syntactically incorrect SQL statement was programmed. **Error correction** Edit the NC program.
160-0069	**Error message** Feed rate is missing **Cause of error** No feed rate was programmed, or the feed rate 0 was programmed. **Error correction** Edit the NC program.

Error number	Description
160-006B	**Error message** Feed rate for tool is missing **Cause of error** In an NC block you programmed F AUTO, but you did not program a feed rate in the TOOL CALL block. **Error correction** - Enter the feed rate directly in the NC block - Program a TOOL CALL block with feed rate
160-0073	**Error message** SQL handles at program end not yet released **Cause of error** A program was ended although table access was still active. **Error correction** Before closing the program, end all SQL table access with SQL COMMIT or SQL ROLLBACK.
160-0082	**Error message** Axis double programmed **Cause of error** You programmed an axis twice in a single positioning block. **Error correction** Edit the part program.
160-0083	**Error message** Axis double programmed **Cause of error** You programmed an axis twice in the Mirror Image cycle. **Error correction** Edit the part program.
160-0084	**Error message** Axis double programmed **Cause of error** While defining Cycle 26 (axis-specific scaling factor) you programmed the scaling factor or the scaling datum twice in one axis. **Error correction** Edit the part program.

Error number	Description
160-0085	**Error message** Axis double programmed **Cause of error** A axis was double-programmed in an approach or departure block. **Error correction** Edit the NC program.
160-0086	**Error message** Axis double programmed **Cause of error** An axis was double-programmed in a coordinate transformation cycle. **Error correction** Edit the NC program.
160-0087	**Error message** Axis double programmed **Cause of error** In a "TCH PROBE" probing cycle an axis was double programmed. **Error correction** Edit the NC program.
160-0089	**Error message** Incorrect number for FN17/FN18 **Cause of error** The number combination for the system data (FN17/FN18) is not allowed. **Error correction** Check the number and index of the system data.
160-008A	**Error message** Incorrect axis index for FN17/FN18 **Cause of error** An incorrect axis index was specified while reading from/writing to system data (FN17/FN18). **Error correction** Check the index of the system date.
160-008B	**Error message** Block format incorrect **Cause of error** A radius was programmed at an illegal location. **Error correction** Edit the NC program.

Error number	Description
160-008C	**Error message** Block format incorrect **Cause of error** A direction of rotation was programmed at an illegal location. **Error correction** Edit the NC program.
160-008F	**Error message** Block format incorrect **Cause of error** Too many axis positions programmed in the NC block. **Error correction** Edit the NC program.
160-0091	**Error message** Block format incorrect **Cause of error** In the NC block APPR/DEP more than 3 axis positions were programmed. **Error correction** Edit the NC program.
160-0092	**Error message** Block format incorrect **Cause of error** In the FN29 block more than 8 values programmed. **Error correction** Edit the NC program.
160-0093	**Error message** Block format incorrect **Cause of error** In a datum shift cycle you programmed too many positions. **Error correction** Edit the NC program
160-0094	**Error message** Block format incorrect **Cause of error** Too many positions programmed in the axis-specific scaling cycle. **Error correction** Edit the NC program.

Error number	Description
160-0095	**Error message** Block format incorrect **Cause of error** In the axis-specific scaling factor cycle you programmed too many center coordinates. **Error correction** Edit the NC program
160-0096	**Error message** Block format incorrect **Cause of error** Too many axes programmed in the mirroring cycle. **Error correction** Edit the NC program.
160-0099	**Error message** Column already assigned to a parameter **Cause of error** In the NC program, the SQL BIND command was used to assign a column name more than once to a parameter. **Error correction** Edit the NC program.
160-009A	**Error message** Table column is not connected with a parameter **Cause of error** Before a table column can accept SQL UPDATE commands (SQL "SELECT...", SQL UPDATE, SQL FETCH), it must be connected with a value by SQL BIND, SQL SYSBIND or by corresponding configuration data. **Error correction** Edit the NC program.
160-009B	**Error message** Column not defined **Cause of error** Description missing for a table column. **Error correction** Check the definition of the table.

Error number	Description
160-009C	**Error message** Block format incorrect **Cause of error** The FN function programmed in the NC program is not supported. **Error correction** Edit the NC program.
160-009E	**Error message** Wrong axis programmed **Cause of error** Incorrect value for FN17:SYSWRITE ID 212 **Error correction** Edit the NC program.
160-009F	**Error message** Block format incorrect **Cause of error** Too many axes were programmed for setting a preset value. **Error correction** Edit the NC program.
160-00A0	**Error message** Spindle is not rotating **Cause of error** A machining cycle was called although the spindle is stopped. **Error correction** Edit the NC program.
160-00A1	**Error message** Incorrect tool index programmed **Cause of error** Invalid tool index programmed. **Error correction** Edit the NC program.
160-00A2	**Error message** Block format incorrect **Cause of error** Faulty workpiece blank definition **Error correction** Edit the NC program.

Error number	Description
160-00A3	**Error message** Axis double programmed **Cause of error** One axis was programmed twice in the workpiece blank definition. **Error correction** Edit the NC program.
160-00A4	**Error message** Wrong axis programmed **Cause of error** Invalid axis in the workpiece blank definition. **Error correction** Edit the NC program.
160-00A5	**Error message** Block format incorrect **Cause of error** The specified sytax element is not allowed in the NC block. **Error correction** Edit the NC program.
160-00A6	**Error message** Cycle is not installed **Cause of error** The programmed cycle is not installed. **Error correction** Check the installed cycles and edit the NC program.
160-00A7	**Error message** System error in interpreter **Cause of error** Internal software error. **Error correction** Inform your service agency.
160-00A8	**Error message** Recursive label call **Cause of error** Within a subprogram, you tried to call the label with which the subprogram begins. **Error correction** Correct the NC program. A subprogram cannot call itself.

Error number	Description
160-00A9	**Error message** Unsuitable touch probe **Cause of error** The desired measuring cycle cannot be performed with the currently selected probe. **Error correction** Edit the NC program.
160-00AA	**Error message** Excessive program nesting **Cause of error** Program nesting through CALL LBL or CALL PGM is too deep, presumably because of a recursive call. **Error correction** Edit the NC program.
160-00AB	**Error message** Recursive program call **Cause of error** Within an NC program, you tried to call the NC program itself as a subprogram. Maybe you tried to call one of the NC programs that called the current NC program. **Error correction** Correct the NC program. An NC program cannot call itself.
160-00AC	**Error message** Not possible on this machine **Cause of error** The desired command cannot be performed on this machine, or the configuration is faulty. **Error correction** Inform the machine tool builder or (if the error occured in an NC program), edit the NC program.
160-00AD	**Error message** Block format incorrect **Cause of error** You programmed an invalid syntax element within a contour definition. Only traversing commands are allowed with the exception of APPR/DEPT and Q parameter calculation. **Error correction** Edit the part program.

Error number	Description
160-00AE	**Error message** Invalid value **Cause of error** You attempted to assign an illegal value to a variable. **Error correction** Edit the NC program.
160-00AF	**Error message** Undeclared string variable **Cause of error** You used a string variable without first declaring it. **Error correction** Edit the NC program. Before it's first use, every string variable must be declared with DECLARE STRING.
160-00B0	**Error message** Block format incorrect **Cause of error** You concluded a contour definition improperly. **Error correction** Edit the NC program. Contours defined by label must end with label 0.
160-00B1	**Error message** Cannot write **Cause of error** You attempted to use FN17:SYSREAD or SQL SYSBIND and SQL FETCH to write to a read-only system parameter. **Error correction** Edit the NC program.
160-00B2	**Error message** String too long **Cause of error** In a table you attempted to write a string that is too long for the corresponding column. **Error correction** Edit the NC program.
160-00B3	**Error message** Block format incorrect **Cause of error** A mandatory syntax element is missing in the NC block. **Error correction** Edit the NC program.

Error number	Description
160-00B4	**Error message** Format file defective **Cause of error** The format file for FN16: F-PRINT (DIN/ISO: D16) has the wrong format. **Error correction** Correct the format file.
160-00B5	**Error message** Format file defective **Cause of error** The format file for FN16: F-PRINT (DIN/ISO: D16) has the wrong format: The Q parameters and other keywords in the specified lined do not fit to the replacement characters in the format string. Remember: - If an %% character in the format string is not to be understood as format information, you must write \%%. - Each line must begin with all keywords that generate an output (such as HOUR, Q14, ...), only then followed by those that do not generate an output (such as M_CLOSE). **Error correction** Correct the format file.
160-00B6	**Error message** Format file defective **Cause of error** The format file for FN16: F-PRINT (DIN/ISO: D16) has the wrong format: The specified line contains the keywords for several different languages. **Error correction** Correct the format file. Each line can contain no more than one keyword for a language.
160-00B7	**Error message** Format file defective **Cause of error** The format file for FN16: F-PRINT (DIN/ISO: D16) has the wrong format: The specified line contains an unknown code word. **Error correction** Correct the format file.

Error number	Description
160-00B8	**Error message** Format file defective **Cause of error** The text generated by FN16: F-PRINT (ISO: D16) is too long. Maximum permissible length: 1024 characters. **Error correction** Change the format file. If necessary, divide the output over several FN16 commands.
160-00B9	**Error message** %1 **Cause of error** **Error correction**
160-00BA	**Error message** File access not possible **Cause of error** You tried to access a file that is reserved for the control manufacture or machine tool builder. **Error correction** Edit the NC program.
160-00BB	**Error message** Faulty CFGREAD **Cause of error** Failed attempt to read a configuration datum via CFGREAD. The desired configuration datum may not exist or it has another type. **Error correction** Check that the name (TAG), key (KEY) and desired attribute (ATTR) is written correctly and that the desired datum has the correct type: In numerical formulas, enter only datum that can be converted to a numerical value (numbers and Boolean variables). In string formulas, enter only data that can be converted to a string (string, Boolean and enumeration).
160-00BD	**Error message** Block format incorrect **Cause of error** A coordinate is required in the specified NC block, but you did not enter one. **Error correction** Edit the NC program.

Error number	Description
160-00BE	**Error message** Illegal probe type **Cause of error** An invalid number was entered for the touch probe model. The probe data or a cycle are probably in error. **Error correction** Correct the type of the touch probe (if necessary through your customer service).
160-00BF	**Error message** CYCL DEF faulty **Cause of error** The Q parameters entered in a cycle as calling parameters are contradictory. The cycle is probably incorrectly defined, or maybe you simply programmed it incorrectly. **Error correction** Correct the NC program with the NC editor. If this does not remedy the error, contact your machine tool builder.
160-00C0	**Error message** %1 **Cause of error** An error was provoked by the function FN14 (ISO: D14) in an NC program or cycle. **Error correction** Look for a description of the error in the machine manual. Then fix the error and restart the program.
160-00C1	**Error message** FN14 error without text **Cause of error** With the FN14 (ISO: D14) function you provoked an error in the NC program or cycle. The string variable to be shown as an error text does not exist. **Error correction** Correct the FN14 function in the NC program. Inform the machine manufacturer if an error message is generated from a cycle.

Error number	Description
160-00C2	**Error message** FN14 error without text **Cause of error** With the FN14 (ISO: D14) function you provoked an error in the NC program or cycle. The additional information is missing for the given error number. **Error correction** Correct the FN14 function in the NC program. Inform the machine manufacturer if an error message is generated from a cycle.
160-00C3	**Error message** M function not permitted **Cause of error** An M function was programmed with a number that is not permitted on this control. **Error correction** Correct the number of the M function.
160-00C4	**Error message** No program selected **Cause of error** An attempt was made to call an NC program with CALL SELECTED, although no program was selected. **Error correction** Correct the NC program.
160-00C6	**Error message** Incorrect tool data **Cause of error** Incorrect tool data **Error correction** Correct the tool table.
160-00C7	**Error message** Invalid value **Cause of error** An invalid value was entered for a parameter in a function. **Error correction** Correct the NC program.

Error number	Description
160-00C8	**Error message** Cycle defective **Cause of error** An incorrect system jump address was entered in a cycle. **Error correction** Inform your machine tool builder.
160-00C9	**Error message** Invalid value for table **Cause of error** You tried to write an invalid value in an SQL table. **Error correction** Correct the NC program.
160-00CA	**Error message** Invalid SQL handle **Cause of error** An invalid SQL handle was entered for the transaction. Maybe the transaction was never successfully opened, or it was already ended through COMMIT or ROLLBACK. **Error correction** Correct the NC program.
160-00CB	**Error message** Error when opening a selected file **Cause of error** Error while opening a file. **Error correction** Ensure that the file exists, that the given path is correct, and that the file has a readable format.
160-00CC	**Error message** Error when opening a system file **Cause of error** Error while opening a system file. **Error correction** Inform your service agency.
160-00CD	**Error message** Block format incorrect **Cause of error** In a NC program you used a syntax element that is allowed only within a contour-definition program. **Error correction** Create a contour definition program and select with SEL CONTOUR.

Error number	Description
160-00CE	**Error message** ID for FN17/FN18 not available **Cause of error** The given ID in the system datum (FN17/FN18) is not available for this channel because the corresponding configuration data are missing. **Error correction** Check the ID of the system datum or run the NC program in another channel. If you really do need a system datum with this ID in this channel, ask your service agency.
160-00CF	**Error message** NC command not available **Cause of error** The given command is not available because the corresponding configuration data are missing. It is probably not supported on this machine. **Error correction** Change the NC program or inform your machine tool builder.
160-00D0	**Error message** M128 / M129 not allowed here **Cause of error** No switch-over of the TCPM is possible while the radius compensation is active (RR/RL, or G41/G42). **Error correction** Activate or deactivate TCPM before beginning or after end of radius compensation.
160-00D1	**Error message** 3-D tool compensation incorrect **Cause of error** LN blocks (face milling) are not possible together with simple radius compensation. **Error correction** If you want face milling, switch off the radius compensation. If you want peripheral milling, switch on TCPM (M128).
160-00D2	**Error message** Only allowed in subprogram **Cause of error** The syntax used is allowed only in subprograms that are called with CALL PGM and in cycles, not in the main program. **Error correction** Correct the NC program.

Error number	Description
160-00D3	**Error message** Variable in calling program not defined **Cause of error** In a subprogram, you tried to change a variable that is not declared in the calling program. **Error correction** Correct the NC program.
160-00D4	**Error message** Tool axis of touch probe not defined **Cause of error** You called a probing cycle without first defining the tool axis of the touch probe. **Error correction** Run the TOOL CALL with the corrent tool axis.
160-00D5	**Error message** OEM system cycle '%3' not defined **Cause of error** You tried to call a non-configured OEM system cycle. **Error correction** Complete the missing OEM system cycle configuration, or correct the NC program containing the call.
160-00D7	**Error message** Inconsistent status of the SQL server **Cause of error** Open transactions to the SQL server exist in the interpreter, even though they have already been closed. **Error correction** Check if the data in the SQL tables matches your expectations.
160-00D8	**Error message** Option not enabled **Cause of error** The function programmed is not allowed on this control or is available only as an option. **Error correction** - Enable the option - Correct the NC program

Error number	Description
160-00D9	**Error message** Only allowed in cycle **Cause of error** The syntax used is allowed only in cycles, not in main programs or subprograms. **Error correction** Correct the NC program.
160-00DA	**Error message** Access to PLC variable has failed **Cause of error** The attempt to access a PLC variable has failed. It may be that the desired variable does not exist. **Error correction** Check that the symbolic name of the variable is spelled correctly and that the variable is defined in the PLC.
160-00DB	**Error message** Access to PLC variable has failed **Cause of error** The attempt to access a PLC variable has failed because it is not of the expected type. **Error correction** Edit the NC program.
160-00DE	**Error message** Program was edited **Cause of error** The current NC program was changed, or one of the NC programs that have called the current NC program. It is therefore not possible to go back into the program. **Error correction** Use the GOTO function or the mid-program startup function to select the desired location to return to in the program.
160-00DF	**Error message** R+ (G43) or R- (G44) not allowed here **Cause of error** The radius compensation R+ or R- cannot be used during active RR or RL. **Error correction** Edit the NC program.

Error number	Description
160-00E0	**Error message** FZ was programmed in conjunction with the number of teeth 0 **Cause of error** You used FZ to define a feed per tooth, although no tooth number is defined in the tool table. **Error correction** Add to the CUT column in the tool table for the active tool.
160-00E1	**Error message** Combination of FU/FZ with M136 not allowed in inch programs **Cause of error** You tried to use FU or FZ (feed rate in inch/rev or inch/tooth) in conjunction with M136 (feed rate in 0.1 inch/rev). **Error correction** Edit the NC program.
160-00E2	**Error message** PGM CALL not allowed **Cause of error** Calling subprograms over CALL PGM is not allowed in the Positioning with MDI mode of operation. **Error correction** Edit the NC program.
160-00E3	**Error message** Spindle speed S is greater than NMAX from the tool table **Cause of error** You entered a spindle speed S greater than the maximum speed defined for this tool in the tool table. **Error correction** - Enter a lower spindle speed S. - Change the maximum spindle speed in the NMAX column of the tool table.
160-00E4	**Error message** Probe cycle started with stylus already deflected **Cause of error** You tried to start a probing cycle although the stylus is still deflected. **Error correction** Increase the retraction path

Error number	Description
160-00E5	**Error message** Axis positions could not be determined **Cause of error** The position of the touch point could not be read. **Error correction** Inform your service agency
160-00E6	**Error message** Missing tool data! **Cause of error** The data for the current tool are not available; the tool is not in the table. Do not run any programs with this condition! **Error correction** - Ensure that the desired tool is entered in the tool table. If required, correct the tool table. - Acknowledge the error message - Conduct a TOOL CALL with an existing tool
160-00E7	**Error message** Incremental input not allowed **Cause of error** In the PATTERN DEF block, you used incremental coordinates to define the first position or a position after a pattern definition. **Error correction** In PATTERN DEF, always use absolute coordinates to program - the first position, - the next position after a pattern
160-00E8	**Error message** Point table has been changed **Cause of error** A pocket table defined with SEL PATTERN has changed during the program run. This is not allowed because the geometry is calculated ahead of time and the changes can then sometimes no longer be considered. **Error correction** Restart the program
160-00E9	**Error message** M function not allowed in a CYCL CALL PAT block **Cause of error** In CYCL CALL PAT block an M function was programmed that is not allowed there. **Error correction** Edit the NC program

Error number	Description
160-00EA	**Error message** CYCL CALL PAT not possible with selected cycle **Cause of error** The selected cycle is not allowed for execution with CYCL CALL PAT. **Error correction** Edit the NC program
160-00EB	**Error message** Recursive call of CYCL CALL PAT **Cause of error** Another CYCL CALL PAT block was found within a point pattern operation. **Error correction** Correct the NC program
160-00EC	**Error message** Radius compensation not allowed before CYCL CALL PAT **Cause of error** The control cannot approach the points of a pattern and compensate the tool radius. Radius compensation must not be switched on before a CYCL CALL PAT block or at the end of the cycle called by this block. **Error correction** Edit the NC program
160-00ED	**Error message** Point pattern call or definition faulty **Cause of error** A CYCL CALL PAT or PATTERN DEF block could not be executed because: - The machining plane programmed with TOOL CALL is not supported or - The point table or a pattern definition with PATTERN DEF is faulty. **Error correction** Check the NC program or point table and correct if necessary
160-00EE	**Error message** No point pattern defined **Cause of error** CYCL CALL PAT can only work if beforehand a point pattern was selected with SEL PATTERN or PATTERN DEF. **Error correction** Edit the NC program

Error number	Description
160-00EF	**Error message** M136 not allowed **Cause of error** M136 is not allowed in combination with the selected cycle. **Error correction** Correct the NC program
160-00F0	**Error message** Faulty call of READ KINEMATICS or WRITE KINEMATICS **Cause of error** You programmed the READ KINEMATICS or WRITE KINEMATICS command incorrectly. Possible causes: - The character string entered under KEY contains an unknown element or an incorrect number of elements - Writing the programmed data to the machine kinematics is not allowed or is not supported **Error correction** Edit the NC program
160-00F1	**Error message** FUNCTION PARAXCOMP or PARAXMODE is faulty **Cause of error** You programmed FUNCTION PARAXCOMP DISPLAY or FUNCTION PARAXCOMP MOVE without indicating an axis, or in FUNCTION PARAXMODE you entered fewer than three axes. **Error correction** - Enter at least one axis in FUNCTION PARAXCOMP DISPLAY or FUNCTION PARAXCOMP MOVE - Enter exactly three axes in FUNCTION PARAXMODE - Correct the NC program
160-00F2	**Error message** M118/M128 at the same time is not allowed **Cause of error** You tried to use M118 to activate a handwheel superimposition during active TCPM. **Error correction** M118 and M128 must not be active at the same time. Edit the NC program.

Error number	Description
160-00F3	**Error message** M118/M128 at the same time is not allowed **Cause of error** You tried to use M118 to activate a handwheel superimposition during active TCPM. The TNC has deactivated the handwheel superimpositioning. **Error correction** M118 and M128 must not be active at the same time. Edit the NC program.
160-00F4	**Error message** Active program was changed **Cause of error** You have changed the active NC program and not yet saved it. **Error correction** Save the unsaved NC program and restart.
160-00F5	**Error message** Called program was changed **Cause of error** You have edited and not yet saved an NC program that was called by another program during program run. **Error correction** Save the called NC program and restart the main program.
160-00F6	**Error message** NC block not allowed in milling mode **Cause of error** You programmed an NC block that is not permitted in milling mode. **Error correction** Correct the NC program.
160-00F7	**Error message** NC block not allowed in turning mode **Cause of error** You tried to execute a function that is not allowed in turning mode. - Fixed cycles for cylindrical surface - Fixed cycles that are locked for turning mode - These manual operation functions are not allowed for turning **Error correction** If necessary, correct the NC program.

Error number	Description
160-00F8	**Error message** Switching to milling or turning operation not allowed **Cause of error** You tried to switch between milling and turning mode while tool radius compensation was active. **Error correction** Cancel the tool radius compensation before switching to milling or turning mode.
160-00F9	**Error message** FUNCTION TURNDATA not allowed on active tool **Cause of error** You tried to use FUNCTION TURNDATA to activate a turning-tool compensation for a milling cutter. **Error correction** Call a turning tool before programming a FUNCTION TURNDATA.
160-00FA	**Error message** M136 not allowed **Cause of error** You attempted with active M136 to make an automatic swing-in movement in connection with Cycle 19 or the PLANE function. **Error correction** Deactivate M136 before swing-in.
160-00FB	**Error message** Spindle speed programmed without configured spindle **Cause of error** You tried to program a spindle speed, although no spindle is configured for the active kinematic configuration. **Error correction** Select a kinematic configuration with spindle, or edit the NC program.
160-00FC	**Error message** 0 rpm not permitted **Cause of error** You called a Rigid Tapping or Tapping cycle with a programmed spindle speed of 0. **Error correction** Program a spindle speed greater than 0.

Error number	Description
160-00FD	**Error message** Feed rate FT or FMAXT not allowed **Cause of error** You defined a feed rate with FT or FTMAX although that is not allowed in connection with APPR LN, LT, CT, PLN, PLT, PCT. **Error correction** Use other feed rate definitions than FT or FTMAX
160-00FE	**Error message** Global Q parameter not defined **Cause of error** A program run (test run) showed that a globally effective Q parameter is not defined. **Error correction** Check whether all required global parameters are in the program head and add more if required.
160-00FF	**Error message** M function not allowed in a CYCL CALL POS block **Cause of error** In the CYCL CALL POS block you programmed an M function that is not allowed there. **Error correction** Edit the NC program
160-0100	**Error message** CYCL CALL POS not allowed with active cycle **Cause of error** The active machining cycle is not allowed in combination with CYCL CALL POS. **Error correction** Call the cycle with M99 or CYCL CALL
160-0101	**Error message** CYCL CALL POS with radius compensation not allowed **Cause of error** You defined a CYCL CALL POS block even though tool-radius compensation is active. **Error correction** Cancel the tool-radius compensation before the CYCL CALL POS block

Error number	Description
160-0102	**Error message** CYCL CALL PAT or CYCL CALL POS: No working direction defined **Cause of error** You did not define a unique working direction in a cycle definition. **Error correction** Check the recent cycle definitions (parameters Q201, Q249, Q356 or Q358).
160-0103	**Error message** Axis angle not equal to tilt angle **Cause of error** - Datum setting inactive with tilted working plane: the position of the tilting axes is not 0°. - Datum setting active with tilted working plane: the position of the tilting axes does not agree with the active angle values. **Error correction** - Move the tilted axes into the default position. - Move the tilting axes to the correct position or adapt the angular values of the tilted axes' position.
160-0104	**Error message** Configuration data cannot be loaded or unloaded **Cause of error** This function is not supported in the Test Run operating mode or by the editor graphics. **Error correction**
160-0105	**Error message** Preset not possible with current machine state **Cause of error** The active preset contains at least one value unequal to zero, which is not allowed because of the present machine condition. The control did not activate this preset. **Error correction** Check the current preset and change if required.

Error number	Description
160-0106	**Error message** NC function not permitted **Cause of error** The block scan evaluated an NC function that cannot be used with mid-program startup (e.g. M142, M143). **Error correction** - If possible, remove the NC function from the program and then restart the block scan. - If this is a control program for calling various part programs, start the desired part program individually in the block scan.
160-0107	**Error message** Impermissible compensation for turning tools **Cause of error** You programmed M128 (TCPM) together with an RL (G41)/ RR (G42) radius compensation or an LN block. This function is not supported in connection with turning tools. **Error correction** Edit the NC program.
160-0108	**Error message** 0 rpm not allowed **Cause of error** You programmed a spindle speed of 0. **Error correction** Always define S to be greater than 0.
160-0109	**Error message** 0 rpm not allowed **Cause of error** Turning mode is active: - You programmed a spindle speed of 0 with FUNCTION TURNDATA SPIN. - You programmed a constant surface speed VC with FUNCTION TURNDATA SPIN, but did not define the cutting speed VC. - You programmed a constant rotational speed (VCONST:OFF) with FUNCTION TURNDATA, but did not define the rotational speed S. **Error correction** - Always program S to be greater than 0. - For constant surface speed (VCONST:ON), always define a surface speed VC. - For constant rotational speed (VCONST:OFF), always define a rotational speed S.

Error number	Description
160-010A	**Error message** Number of program part repeats is defined as 0 **Cause of error** In a CALL LBL statement (DIN/ISO: L x,y), you defined the number of program part repeats with 0. **Error correction** - Define a number of repeats between 1 and 9999. - Define a subprogram call without entering repetitions.
160-010B	**Error message** No turning tool is active **Cause of error** The TNC cannot execute the function with the data of the active tool. **Error correction** - Insert a turning tool. - Check the TYPE column of the lathe tool table.
160-010C	**Error message** Contradictory tool data **Cause of error** You tried to call a lathe tool with data that are allowed only for milling tools, e.g. - Spindle speed S - Oversizes DL, DR, DR2 **Error correction** Correct the NC program.
160-010D	**Error message** Tool data are contradictory during active turning operation **Cause of error** The following functions are not allowed during active turning operation (FUNCTION MODE TURN): - Definition of the spindle speed for the tool spindle (TOOL CALL S, DIN/ISO: S) - Definition of the tool axis direction (e.g. TOOL CALL Z, DIN/ISO: G17/G18/G19) **Error correction** Correct the NC program.
160-010E	**Error message** Incorrect file type **Cause of error** The type of entered data is not permissible in this application. **Error correction** Please check whether the file has the correct extension.

Error number	Description
160-010F	**Error message** Spline interpolation is not supported **Cause of error** You tried to run an NC block that was described as a spline. This block format is not supported by the control. **Error correction** Select an NC program without spline interpolation or edit the NC program.
160-0110	**Error message** FN27/FN28: No table selected **Cause of error** You attempted to use FN27 to write to a table, or FN28 to read from a table, although no table was open. **Error correction** Use FN26 to open the desired table.
160-0111	**Error message** FN27/FNF28: Field is not numerical **Cause of error** You attempted to use an FN27 or FN28 function to write to or read from a non-numerical field. **Error correction** Writing and reading operations are possible only with numerical fields.
160-0112	**Error message** MDI: Radius compensation not possible **Cause of error** No tool radius compensation is allowed in MDI mode. **Error correction** Correct the NC block
160-0113	**Error message** Label is missing Label is missing **Cause of error** You used Cycle 14 to call a nonexistent label. **Error correction** Change the number or name in Cycle 14 or insert the missing label

Error number	Description
160-0114	**Error message** Label is missing Label is missing **Cause of error** You used FUNCTION TURNDATA BLANK LBL to call a nonexistent label. **Error correction** Change the number or name in FUNCTION TURNDATA BLANK LBL or insert the missing label
160-0115	**Error message** Too many vertical profiles programmed **Cause of error** The selected surface profile contains too many vertical profiles. **Error correction** Edit the NC program
160-0116	**Error message** Type of interpolation invalid **Cause of error** No interpolation type or an unsupported interpolation type was programmed. **Error correction** Edit the NC program
160-0117	**Error message** Too many axes programmed **Cause of error** You programmed too many axes for paraxial interpolation. **Error correction** Edit the NC program
160-0118	**Error message** Plane profile incorrectly defined **Cause of error** The selected contour cannot be used as the plane profile of a profile surface. The plane profile must consist only of a subcontour. **Error correction** Edit the NC program

Error number	Description
160-0119	**Error message** Circle calculation failed **Cause of error** It was not possible to calculate a circle from the given coordinates. **Error correction** Edit the NC program
160-011A	**Error message** No tool touch probe active **Cause of error** You tried to switch a tool touch probe without first activating it. **Error correction** Activating the tool touch probe.
160-011B	**Error message** At least one parameter within the contour is not initialized **Cause of error** The QL parameters of this program cannot be accessed in the contour defined in the current program. **Error correction** Define the QL parameters used within the contour definition or use globally effective Q parameters.
160-011F	**Error message** Range of traverse not defined **Cause of error** You tried to switch to a traverse range that was not defined in the configuration through CfgWorkingRange. **Error correction** Inform your machine tool builder
160-0120	**Error message** DATA ACCESS has failed **Cause of error** An incorrect path was used for DATA ACCESS or there are no access rights. **Error correction** - Correct the programmed path for DATA ACCESS - Move the NC program to the PLC partition

Error number	Description
160-0121	**Error message** Label is missing **Cause of error** A workpiece blank definition (NC block: BLK FORM ROTATION) refers to a nonexistent label. **Error correction** - Correct the NC program: Change the reference to the label or insert the missing label.
160-0122	**Error message** Contradictory M functions **Cause of error** In the same block, you tried to output multiple effective M functions that cancel each other. For example, you cannot program M3 and M4 in the same NC block because both go into effect at beginning of the block. On the other hand, M3 and M5 are allowed in the same block because M3 goes into effect at the block beginning and M5 at block end. The machine tool builder specifies which M functions cancel each other. **Error correction** Correct the NC program.
160-0123	**Error message** Alias strobe of type %1 not configured **Cause of error** The entered command is not available because the corresponding configuration data are missing. Presumably it is not supported on this machine. The config object CfgPlcStrobeAlias is missing or has not been entered in the CfgPlcStrobes/aliasStrobes list. **Error correction** Edit the NC program or contact your machine tool builder.
160-0124	**Error message** CFGWRITE incorrect **Cause of error** The attempt to use CFGWRITE to write a configuration datum has failed. It might have another type, or the the write axis is not allowed. **Error correction** Check the spelling of TAG, KEY, attribute (ATR) and the type of value to be written (DAT). If required, move the program from the TNC to the PLC partition.

Error number	Description
160-0125	**Error message** Demo version **Cause of error** This software is a demo version. With the demo version you can edit and run NC programs with a maximum length of 100 lines. With the demo version you are not entitled to service support from HEIDENHAIN. **Error correction** - If you would like to purchase the licensed version of the product, please contact the service agency for the control. - Check whether the keyboard of the programming station (or the dongle) is connected.
160-0126	**Error message** M90 is not supported by this NC software **Cause of error** The M90 function is no longer available with this NC software. **Error correction** Use Cycle 32 TOLERANCE in order to influence accuracy, surface quality and machining speed.
160-0127	**Error message** M105 and M106 are not supported with this NC software **Cause of error** The M105 and M106 function is not available with this NC software. **Error correction**
160-0128	**Error message** M104 is not supported with this software **Cause of error** M104 is not supported by this NC software. The datum set manually most recently is saved in row 0 of the preset table. **Error correction** - Use Cycle 247 SET DATUM to activate a preset from the preset table.
160-0129	**Error message** M112 and M113 are not supported with this NC software **Cause of error** The M115 and M113 functions are not available with this NC software. **Error correction** Use Cycle 32 TOLERANCE in order to influence accuracy, surface quality and machining speed.

Error number	Description
160-012A	**Error message** M114 and M115 are not supported with this NC software **Cause of error** M114 und M115 are not supported by this NC software. **Error correction** - Use M144/145 or M128/129 in order to take the positions of rotary and tilting axes into consideration.
160-012B	**Error message** M124 is not supported with this NC software **Cause of error** M124 is not supported by this NC software. The contour filter cannot be programmed in the NC program. **Error correction** - Use CfgStretchFilter to configure the contour filter in the machine configuration.
160-012C	**Error message** M132 is not supported with this NC software **Cause of error** M132 is not supported by this NC software. **Error correction** - Use Cycle 32 TOLERANCE in order to influence accuracy, surface quality and machining speed.
160-012D	**Error message** M134 and M135 are not supported with this NC software **Cause of error** This NC software does not support M134 and M135 for a precision stop at non-tangential transitions for positioning moves with rotary axes. **Error correction** - Use machine-specific funtions to activate or deactivate automatic clamping. - If required, contact your machine tool builder
160-012E	**Error message** M142 is not supported with this NC software **Cause of error** The deletion of modal program functions through M124 is not supported by this NC software. **Error correction**

Error number	Description
160-012F	**Error message** M150 is not supported with this NC software **Cause of error** This NC software does not support the suppression of limit-switch messages through M150. **Error correction**
160-0130	**Error message** M200 to M204 are not supported with this NC software **Cause of error** This NC software does not support M200-M204 for laser cutting. **Error correction**
160-0136	**Error message** SL1 cycles are not supported with this NC software **Cause of error** The SL1 cycles 6, 15 and 16 are not supported by this NC software. **Error correction** - Adapt the NC program: Replace SL1 cycles by the new, improved SL2 cycles 20, 21, 22, 23 and 24.
160-0137	**Error message** The touch probe cycle is not supported with this NC software **Cause of error** The programmed touch probe cycle is not supported by this NC software. **Error correction** - Adapt the NC program: Replace the touch probe cycle 2 or 9 by the newer cycles 461, 462 or 463.
160-0138	**Error message** Cycle 30 is not supported with this NC software **Cause of error** Cycle 30 is not supported by this NC software. **Error correction**

Error number	Description
160-0139	**Error message** Tool must not be changed Tool must not be altered **Cause of error** A change of tool number or tool index is not allowed at present. This test was activated by the machine tool builder within the TOOL CALL macro. **Error correction** Inform your machine tool builder
160-013A	**Error message** Mid-program startup to area with active RTC function not possible **Cause of error** You tried to run a mid-program startup on an area in the NC program in which the RTC function (Real-Time Coupling) is active. This is not possible. **Error correction** - Adapt the target position for the block search - Choose a target position so that the mid-program startup ends before activation of the RTC function.
160-013B	**Error message** Function locked **Cause of error** You have programmed a function that has not been enabled by the machine manufacturer. **Error correction** - Edit the NC program or contact your machine tool builder
160-013C	**Error message** Function not possible with current tool **Cause of error** A function is incompatible with the properties of the current tools. **Error correction** - Edit the NC program - Check the tool data

Error number	Description
160-013D	**Error message** File %1 for tool carrier kinematics could not be opened **Cause of error** The given file for a tool carrier kinematic model could be found neither under TNC:\system\Toolkinematics nor under OEM:\config\Toolkinematics. **Error correction** - Add the file in one of the appropriate directories or delete the corresponding entry under "KINEMATIC" in the tool table if no tool-carrier kinematic model is to be used for the tool. - Inform your service agency.
160-013E	**Error message** Data of the tool carrier kinematics is faulty **Cause of error** The tool-carrier kinematic data for the current tool are faulty. Note: Do not run any NC programs in this condition and be careful if you move the axes manually! **Error correction** - Ensure under "KINEMATICS" that a valid file for the tool-carrier kinematics is entered for the desired tool in the tool table. - Acknowledge the error message - Run a TOOL CALL for a tool that has no tool-carrier kinematics assigned or one that has a valid tool-carrier kinematic model. - Inform your service agency.
160-0142	**Error message** Axis positioning not possible **Cause of error** - An auxiliary axis movement was moved before the last movement of this axis was completed **Error correction** Check the PLC program. - Wait explicitly for the end of the previous positioning movement or cancel it - Inform your service agency.
160-0143	**Error message** Axis movement canceled **Cause of error** The positioning movement of an auxiliary axis was canceled. **Error correction** - If necessary, check other pending error messages - Check the PLC program and status marker of the axis - Inform your service agency

Error number	Description
160-0144	**Error message** The configuration datum %2 is missing for the tool touch probe %1 **Cause of error** The appropriate configuration datum is missing for the TT tool touch probe activated with the CfgProbes/activeTT parameter. **Error correction** Add the missing configuration datum or activate another TT tool touch probe.
160-0145	**Error message** No active touch probe **Cause of error** You started a probing cycle without first activating the touch probe. **Error correction** - Insert the TS touch probe, or - switch to the TT tool touch probe.
160-0146	**Error message** Alias strobe of type %1 is incorrectly configured **Cause of error** The entered command is not available because the corresponding configuration data are faulty. Presumably it is not supported on this machine. The M function referenced in CfgPlcStrobeAlias is not configured. **Error correction** - Inform your machine tool builder
160-0155	**Error message** Ambiguity: DR2 and DR2TABLE defined in the tool table **Cause of error** Ambiguous DR2 data are entered in the tool table. Only DR2 or DR2TABLE can set for a tool at any given time. **Error correction** In order to use 3-D radius compensation, enter the desired DR2 value and leave the DR2TABLE entry empty In order to use the 3D-ToolComp option, enter in DR2TABLE the file name (without extension) of the compensation value table and set DR2 to zero (0)

Error number	Description
160-0156	**Error message** Radius compens. impossible with milling cutter (TCPM TIP-CENTER) **Cause of error** An attempt has been made to program a radius compensation using a milling tool and active TCPM REFPNT TIP CENTER. Radius compensation with this preset is possible only for turning tools. **Error correction** Insert a turning tool.
160-0157	**Error message** Activation of TCPM REFPNT TIP-CENTER is not allowed **Cause of error** An attempt was made to activate TCPM REFPNT TIP-CENTER. This is not allowed in the current program context. TCPM REFPNT TIP-CENTER cannot be used in the following program states: - Active tool radius compensation - Active 3-D tool compensation **Error correction** If necessary, deactivate the active tool radius compensation or 3-D tool compensation
160-0158	**Error message** TCPM TIP-CENTER active **Cause of error** TCPM REFPNT TIP-CENTER is active. The following functions cannot be used with this TCPM reference point: - 3-D tool compensation - Radius compensation R+ and R- **Error correction** Deactivate TCPM TIP-CENTER (using FUNCTION TCPM RESET or M129) or select TIP-TIP or CENTER-CENTER for the TCPM preset before activating the tool compensation.
160-015A	**Error message** TCPM TIP-CENTER is active with tool radius compensation **Cause of error** TCPM REFPNT TIP-CENTER with tool radius compensation is active. The following functions are not possible in this state: - M128 - M129 / FUNCTION TCPM RESET - FUNCTION TCPM REFPNT: Change of TCPM preset **Error correction** Deactivate tool radius compensation first with R0

Error number	Description
160-0160	**Error message** M2/M30 in subprogram **Cause of error** The NC program was ended by a call from M2 or M30 in a subprogram called with CALL PGM. **Error correction** - If this behavior is desired, no further measures are required. - If you want to return from the subprogram, edit the NC program and jump to the last line of the subprogram.
160-0162	**Error message** File with 3D-ToolComp compensation values not available **Cause of error** The file with the name provided in the tool-table column DR2TABLE could not be opened. **Error correction** - Check the spelling of the file name in the tool table. - Check whether the file is located in the intended folder on the control. - Inform your service agency.
160-0163	**Error message** NC command not allowed during tool-oriented machining **Cause of error** - NC command was executed that is not supported with tool-oriented machining. In this mode, for example, no change is allowed of the preset table through traverse-range switchover. **Error correction** - Correct the NC program, or - use workpiece-oriented machining
160-0164	**Error message** Parameter not permissible in macro **Cause of error** In an OEM macro, TOOL CALL was programmed with invalid parameters. - With TOOL CALL, no parameters are allowed in the macro for tool-oriented pallet machining. - In the TOOL CALL, only number, step index and spindle speed are allowed in the tool change macros. **Error correction**

Error number	Description
160-0165	**Error message** Facing slide: M148 not allowed **Cause of error** Automatic tool lift-off not allowed with an active facing slide. **Error correction** - Check the NC program and adapt it if necessary.
160-0166	**Error message** Function locked **Cause of error** The machine tool builder disabled this function with a configuration setting. **Error correction** Edit the NC program or contact your machine tool builder.
160-0167	**Error message** Cannot continue program **Cause of error** In rare cases, a program resumption with GOTO is no longer possible. **Error correction** - Restart the NC program - If required, try a mid-program startup
160-0168	**Error message** Preset not allowed **Cause of error** The activated preset contains at least one value not equal to zero. This is not allowed due to a limitation defined by the machine tool builder. The limitation could have been activated by the following causes: - Globally by configuration - Dependency on a machine condition, through an NC syntax. The control did not activate the impermissible value of this preset. **Error correction** - Check the current preset and change it if required - In the Test Run operating mode, machining preset loading can be aided using the blank-in-workspace function - Inform your service agency

Error number	Description
160-0169	**Error message** OEM offset for X, Y, or Z axis not permissible **Cause of error** You tried to define an OEM offset for one of the principal axes X, Y or Z. That is not allowed. **Error correction**
160-016A	**Error message** Automatic tool change not possible **Cause of error** An automatic tool change is supported at preset only for milling operations. **Error correction** - In the AFC settings, specify a strategy that does not have the overload response OVLD = M (macro). - Contact your service agency if this message did not occur due to an AFC overload reaction.
160-016B	**Error message** Switching the machining mode is not allowed **Cause of error** You tried to switch the operating mode while tool radius compensation was active. **Error correction** Cancel the tool radius compensation before switching the operating mode.
160-016C	**Error message** NC block in grinding mode is not allowed **Cause of error** You tried to execute a function that is not allowed in grinding mode. - Machining cycles for cylindrical surface - Machining cycles that are disabled for grinding mode - Functions of manual operation that are not permitted for grinding **Error correction** - Correct the NC program

Error number	Description
160-016D	**Error message** NC block in dressing mode is not allowed **Cause of error** You tried to execute a function that is not permitted in dressing mode. - Machining cycles for cylindrical surface - Machining cycles that are disabled for dressing mode - Functions of manual operation that are not permitted for dressing **Error correction** - Correct the NC program
160-016E	**Error message** M function not permitted **Cause of error** In the M function dialog you indicated an M function that is only permitted in the NC program. **Error correction** - Correct the number of the M function - Output the function via an MDI block.
160-016F	**Error message** Execution request for NC syntax missing **Cause of error** You do not have the rights to perform this specific function (e.g. FN22). **Error correction** Edit the NC program.
160-0170	**Error message** M89 not supported in combination with FK **Cause of error** It was attempted to execute an FK block while M89 was active. This is not supported. **Error correction** - Edit the NC program. - In order to execute the selected cycle, program M99 for the respective FK blocks instead of M89.
160-0171	**Error message** FN27/FN28 type of field not supported **Cause of error** You tried to use an FN27 or FN28 function to access a field that does not match the type of parameter. **Error correction** - For numerical fields you must use a Q, QL, or QR parameter as the source or target. - A QS parameter must be used for text fields.

Error number	Description
160-0177	**Error message** Grinding tool not defined completely **Cause of error** - You have called a griding tool that is not defined in the griding tool table, or at least not completely. - The griding tool table does not exist or is faulty. **Error correction** - Add the missing tool to the griding tool table. - Create or correct the griding tool table.
160-0179	**Error message** Traverse range not defined **Cause of error** You tried to deactivate a traverse range that is not defined in the configuration through CfgWorkingRange. **Error correction** Inform your machine tool builder.
160-017A	**Error message** Maximum number of configuration changes was exceeded **Cause of error** Too many configuration changes were buffered with WRITE CFG PREPARE. **Error correction** Write the already prepared changes with WRITE CFG COMMIT before any further PREPARE instructions.
160-017B	**Error message** NC program faulty: WRITE CFG COMMIT without PREPARE **Cause of error** You programmed WRITE CFG COMMIT without a preceding WRITE CFG PREPARE, or the most recently programmed WRITE CFG PREPARE is no longer valid. **Error correction** Edit the NC program
160-017C	**Error message** Parameter type is incorrect **Cause of error** The result type in a Q parameter formula does not match the parameter on the left side: - On the left is a Q, QR, or QL parameter but the result of the formula is a string - On the left is a QS parameter but the result of the formula is a number **Error correction** Correct the formula

Error number	Description
160-017D	**Error message** Probing movement was prevented by DCM **Cause of error** The collision monitoring function shortened the length of the probing movement to 0. **Error correction** - Check the configuration of the touch probe being used - Check whether a collision object must be deactivated for the probing operation
160-017E	**Error message** Block scan not permitted **Cause of error** You began a mid-program startup while dressing mode was active or the target of a mid-program startup is at a position in the dressing mode. This is not allowed. **Error correction** Adapt the target position for the block search
160-017F	**Error message** Handwheel superimpositioning not allowed in dressing mode **Cause of error** You attempted to switch to dressing mode even though handwheel superimpositioning is still active. **Error correction** Switch off handwheel superimpositioning before switching to dressing mode.
160-030A	**Error message** No technology data record available for contour machining **Cause of error** A Cycle 271 must be programmed before every fixed cycle 272, 273, or 274. **Error correction** - Adapt the NC program - Program Cycle 271
160-030B	**Error message** Block scan not permitted **Cause of error** The target position of the mid-program startup is within an active reciprocating movement. This is not allowed. **Error correction** Adapt the target position for the block search

Error number	Description
160-030E	**Error message** Preset not defined **Cause of error** An attempt was made to select a preset that is not defined. **Error correction** Correct the name or the preset or expand the preset table.
160-030F	**Error message** File type not permissible for the tool-model file **Cause of error** Für das einzuwechselnde Werkzeug wurde eine Datei angegeben, die keinen zulässigen Dateityp hat. Zulässig sind *.stl-Dateien. **Error correction** Ersetzten Sie die Datei durch eine zulässige Datei.
160-0310	**Error message** File with tool model is missing **Cause of error** Die in der Werkzeugtabelle angegebene Datei zur Beschreibung des Werkzeugmodells ist nicht vorhanden **Error correction** - Schreibweise des Dateinamens in der Werkzeugtabelle kontrollieren - Datei in den dafür vorgesehenen Ordner auf der Steuerung kopieren (Angaben im Benutzerhandbuch beachten)
1A0-0001	**Error message** System error in internal path calculation Invalid message %1 **Cause of error** System error **Error correction** Inform your service agency.
1A0-0002	**Error message** System error in the geometry chain: %1 **Cause of error** System error **Error correction** Inform your service agency.

Error number	Description
1A0-0003	**Error message** Function not yet implemented: %1 **Cause of error** You tried to use a function that is not available in this version of the NC software. **Error correction** Edit the NC program.
1A0-0004	**Error message** System error in the geometry chain: %1 **Cause of error** System error in the geometry chain **Error correction** Inform your service agency.
1A0-0005	**Error message** System error in the geometry chain: %1 **Cause of error** System error in internal path calculation **Error correction** Inform your service agency
1A0-0006	**Error message** System error in the geometry chain: %1 **Cause of error** System error in internal path calculation **Error correction** Inform your service agency
1A0-0007	**Error message** System error in the geometry chain: %1 **Cause of error** System error in the geometry chain **Error correction** Inform your service agency.

Error number	Description
1A0-0008	**Error message** Axis double programmed **Cause of error** In a circle-center block or a pole block (CC, ISO: I,J,K) you programmed the same axis twice. **Error correction** Edit the NC program
1A0-0009	**Error message** Axis in CC block (ISO: I,J,K) double programmed **Cause of error** In a circle-center block or a pole block (CC, ISO: I,J,K) you programmed the same axis twice. **Error correction** Edit the NC program
1A0-000A	**Error message** No tool axis defined **Cause of error** No plane selection or tool axis direction programmed, or no default plane specified. **Error correction** Edit the NC program or specify the default working plane in the configuration.
1A0-000B	**Error message** No arc end point programmed **Cause of error** Arc end point data missing **Error correction** Program at least one coordinate of the arc end point.
1A0-000C	**Error message** Arc block: No direction of rotation programmed **Cause of error** You programmed a circle without a direction of rotation. **Error correction** Always program the direction of rotation (DR).
1A0-000D	**Error message** No arc radius programmed in CR block **Cause of error** Missing arc radius in CR block **Error correction** Program a radius in the CR block.

Error number	Description
1A0-000E	**Error message** System error in the geometry chain: %1 **Cause of error** System error in the geometry chain **Error correction** Inform your service agency.
1A0-000F	**Error message** Pole is missing **Cause of error** You tried to move with polar coordinates (LP/CP/CTP, DIN/ISO: G10/G11/G12/G13/G15/G16), without first programming a pole (CC) (ISO: I/J/K); or you did not first enter a circle center when programming a circle. **Error correction** Program the pole (CC) before the first block with polar coordinates (ISO: I, J; K), or first program a circle center (CC) when programming a circle.
1A0-0010	**Error message** System error in the geometry chain: %1 **Cause of error** System error in the geometry chain **Error correction** Inform your service agency.
1A0-0011	**Error message** Programmed axis not assigned to a physical axis **Cause of error** Axis value programming with programmable axis without assigned physical axis **Error correction** - Correct the NC program. - Inform your service agency.
1A0-0012	**Error message** System error in the geometry chain: Gap in the programmed path **Cause of error** System error in the geometry chain **Error correction** Inform your service agency.

Error number	Description
1A0-0013	**Error message** Tool radius too large **Cause of error** - Inside contour radius smaller than tool radius - Tool compensation results in contour damage (loop in the path of the tool center) - Contour offset too large **Error correction** - Edit the NC program. - Select a smaller tool. - Program a smaller contour offset.
1A0-0016	**Error message** Error in module configuration: %1 **Cause of error** Error in the module configuration (uninitialized list in configuration object) **Error correction** - Edit the configuration data. - Inform your service agency.
1A0-0017	**Error message** Error in module configuration: %1 **Cause of error** Error in module configuration (list size of an attribute in configuration object is too small) **Error correction** - Edit the configuration data. - Inform your service agency.
1A0-0019	**Error message** Switch the diameter programming off before eccentric grinding **Cause of error** Diameter programming not switched off before eccentric grinding **Error correction** Edit the cycle.

Error number	Description
1A0-001B	**Error message** Automatic pole capture not permitted **Cause of error** You programmed (automatic pole capture) a CC block (DIN/ ISO: I,J,K) without coordinates. This is not possible in the current context, since the TNC cannot clearly identify the plane for the pole. **Error correction** - In the block directly before the pole-capture block, program two linear axes of the working plane. - Specify the working plane via TOOL CALL
1A0-001D	**Error message** Incorrect pole axis for selected working plane **Cause of error** - Z component with XY plane - X component with YZ plane - Y component with ZX plane **Error correction** Edit the NC program.
1A0-001E	**Error message** Chamfer/rounding arc: Subsequent line lies in the wrong plane **Cause of error** After programming a transition element (RND/CHF), you programmed a linear element that does not lie in the plane of the transition element. **Error correction** Edit the NC program.
1A0-0021	**Error message** System error in the geometry chain: %1 System error in the geometry chain: %1 **Cause of error** System error in the geometry chain **Error correction** Inform your service agency.

Error number	Description
1A0-0024	**Error message** An axis must not be locked after a contour transition element **Cause of error** Rounding or chamfer programmed immediately before axis locking **Error correction** - Correct the NC program. - Eliminate contour transition element from program, or - Program target position after contour transition element.
1A0-0025	**Error message** Presetting not permitted after a contour transition element **Cause of error** Setting the base transformation immediately after contour transition element **Error correction** - Correct the NC program. - Eliminate contour transition element from program, or - Program target position after contour transition element.
1A0-0026	**Error message** Setting of axis values not permitted immediately after contour transition element. **Cause of error** Setting of axis values immediately after contour transition element **Error correction** - Edit the NC program. - Eliminate contour transition element from program or - Program target position after contour transition element.
1A0-0027	**Error message** Tool change not permitted right after contour transition element **Cause of error** Tool change immediately after contour transition element **Error correction** - Correct NC program. - Eliminate contour transition element from program, or - Program target position after contour transition element.

Error number	Description
1A0-0028	**Error message** Probe block not permitted right after contour transition element **Cause of error** Probing block immediately after contour transition element **Error correction** - Correct the NC program. - Eliminate contour transition element from program, or - Program target position after contour transition element.
1A0-002B	**Error message** Touch point inaccessible **Cause of error** In the TCH-PROBE 0 cycle (ISO: G55) or during use of the manual probing cycles, no touch point was reached within the traverse path defined in the touch probe table. **Error correction** - Preposition the touch probe closer to the workpiece. - Increase the value in the touch probe table.
1A0-002C	**Error message** Two successive APPR/DEP movements programmed **Cause of error** - Two approach/depart movements programmed in succession. - Elements with zero length were programmed between two approach/departure movements. **Error correction** Edit the NC program.
1A0-002D	**Error message** Compensation switching not permitted **Cause of error** Change of compensation without previous end of compensation **Error correction** - Edit the NC program. - Close the previous contour.
1A0-002F	**Error message** Tool definition is missing **Cause of error** The geometry chain has received no tool data before switching on the tool compensation. **Error correction** - Edit the NC program. - Inform your service agency.

Error number	Description
1A0-0030	**Error message** Path comp wrongly started **Cause of error** You tried to activate the tool radius compensation with RL or RR (ISO: G41 or G42) in an arc block. **Error correction** Activate the tool radius compensation only with a linear block (L, ISO: G0, G1, G10, G11).
1A0-0031	**Error message** Path comp wrongly ended **Cause of error** You tried to cancel the tool radius compensation with R0 (ISO: G40) in an arc block. **Error correction** Cancel the tool radius compensation only with a linear block (L, ISO: G0, G1, G10, G11).
1A0-0032	**Error message** Switching the working plane not permitted **Cause of error** - Change of working plane after RND, CHF, APPR **Error correction** - Edit the NC program.
1A0-0033	**Error message** Rounding cannot be calculated **Cause of error** - The rounding was programmed in the first positioning block of the NC program. - Before the rounding, a contour element was programmed that does not lie exclusively in the working plane. - Before the rounding, a contour element was programmed that has no geometric length. - An RND block (DIN/ISO: G25) was selected with GOTO after a program interruption. **Error correction** - At least two contour elements must be programmed before the rounding. - The contour element immediately before the rounding must be programmed exclusively in the working plane. - After a program interruption, restart at least two positioning blocks before the RND block.

Error number	Description
1A0-0034	**Error message** Chamfer cannot be calculated **Cause of error** - The chamfer was programmed in the first positioning block of the NC program. - Before the chamfer, a contour element was programmed that does not lie exclusively in the working plane. - Before the chamfer, a contour element was programmed that has no geometric length. - An CHF block (DIN/ISO: G24) was selected with GOTO after a program interruption. **Error correction** - At least two contour elements must be programmed before the chamfer. - The contour element immediately before the chamfer must be programmed exclusively in the working plane. - After a program interruption, restart at least two positioning blocks before the CHF block.
1A0-0035	**Error message** Two successive transitional elements not permitted **Cause of error** - Two transition elements were programmed in succesion - CHF after transition element - RND after transition element - APPRLT after transition element - APPRLN after transition element - APPRCT after transition element - APPRLCT after transition element - DEPLT after transition element - DEPLN after transition element - DEPCT after transition element - DEPLCT after transition element **Error correction** - Edit the NC program.
1A0-0036	**Error message** Contour transition not concluded **Cause of error** - Missing curve element after CHF/RND - E.g. program end after CHF/RND **Error correction** - Edit the NC program.

Error number	Description
1A0-0037	**Error message** General system error in the geometry chain Internal error in OCM contour milling cycle **Cause of error** - Contradictory data. **Error correction** - Inform your service agency.
1A0-0038	**Error message** Incorrect application of Cycle 19 **Cause of error** - Before Cycle 19 was called, tilted working plane was switched off (FN17 ID210 NR6). **Error correction** Edit the NC program.
1A0-0039	**Error message** Incorrect application of Cycle 19 **Cause of error** - Before Cycle 19 was called, Cycle 8 or Cycle 10 was called. **Error correction** Edit the NC program.
1A0-003A	**Error message** Incorrect axis index for FN18 **Cause of error** - In the Read System Data function (FN18, ISO: D18), you entered an incorrect axis index. **Error correction** - Check the index of the system datum.
1A0-003B	**Error message** System error in the geometry chain: Message in wrong state %1 %2 %3 **Cause of error** - System error in the geometry chain **Error correction** - Inform your service agency.

Error number	Description
1A0-003C	**Error message** Calculation of tool center path or contour-linking operations for present contour lists failed! **Cause of error** - Internal software error **Error correction** - Change the contour definitions - Inform your service agency
1A0-003D	**Error message** System error in the geometry chain: %1 **Cause of error** - System error in the geometry chain **Error correction** - Inform your service agency.
1A0-003E	**Error message** Tool compensation must not be switched on after G27 **Cause of error** An attempt was made to switch on the radius compensation after G27. **Error correction** Remove the radius compensation.
1A0-003F	**Error message** Linear movement not allowed **Cause of error** You tried to program a linear movement after G27 before deactivating the radius compensation. **Error correction** - First switch off the radius compensation
1A0-0040	**Error message** Approach movement by means of RND not allowed right after a contour transition element **Cause of error** An approach was programmed with RND after RND or CHF. **Error correction** - Edit the NC program.

Error number	Description
1A0-0041	**Error message** G26 programmed in wrong context **Cause of error** - Radius compensation was not switched on immediately after G26. **Error correction** - Switch on the radius compensation immediately before G26.
1A0-0042	**Error message** G26 not allowed after corner rounding or chamfer **Cause of error** You programmed a G26 after RND (ISO: G25) or CHF (ISO: G24). **Error correction** - Edit the NC program
1A0-0043	**Error message** G27 not allowed when tool compensation is off **Cause of error** A G27 was programmed, although tool compensation was switched off. **Error correction** - Edit the NC program.
1A0-0044	**Error message** Radius compensation must not be switched on in a block preceding the approach block **Cause of error** You tried to switch on the tool radius compensation before an approach block. **Error correction** - Switch on the tool radius compensation only in the approach block
1A0-0045	**Error message** System error in the geometry chain: %1 %2 **Cause of error** System error: Data in a message are contradictory. **Error correction** - Inform your service agency.

Error number	Description
1A0-0046	**Error message** Pole programming with only one coordinate not allowed **Cause of error** You defined a pole in only one axis. **Error correction** - Program two axes or no axes (automatic pole transfer)
1A0-0047	**Error message** Circular arc after G27 not allowed **Cause of error** A circular arc was programmed after G27. **Error correction** - Edit the NC program.
1A0-0048	**Error message** Helix after corner rounding or chamfer not allowed **Cause of error** A helix was programmed after a contour transition element. **Error correction** - Edit the NC program.
1A0-0049	**Error message** Height of helix must not lie a circle plane **Cause of error** You programmed an incorrect axis in the direction perpendicular to the circle plane. **Error correction** - Edit the NC program
1A0-004A	**Error message** Angle of a helical path must be programmed incrementally **Cause of error** You programmed a helix without incremental data on the angle. **Error correction** - Edit the NC program

Error number	Description
1A0-004B	**Error message** The sign and direction of a circle's angle are contradictory **Cause of error** In a circular movement, you programmed a negative angle increment to a positive direction of rotation (counterclockwise) or a positive angle increment to a negative direction of rotation (clockwise). **Error correction** - Correct the direction of the movement or the algebraic sign of the angle
1A0-004C	**Error message** Tangent at start point of element not defined **Cause of error** A geometrical element with tangential transition (e.g. CT) was programmed. However, the tangent at the start point is undefined, e.g. because of an immediately preceding verticaldeparture. or because it is the 1st block of the NC program. **Error correction** - Edit the NC program.
1A0-004D	**Error message** First geometric element after scaling is not a straight line **Cause of error** A scaling was programmed immediately before a geometric element that is not a linear element. **Error correction** - Edit the NC program.
1A0-004E	**Error message** Linear movement after tangential departure is not in circle plane **Cause of error** Immediately after a tangential departure, a linear element was programmed that contains a component perpendicular to the arc plane. **Error correction** Edit the NC program.
1A0-004F	**Error message** Tool compensation not switched off for 5-axis movement **Cause of error** You programmed a linear 5-axis movement although the tool radius compensation was active. **Error correction** - Switch off the tool radius compensation beforehand

Error number	Description
1A0-0050	**Error message** Linear 5-axis movement not allowed after transition element **Cause of error** A linear 5-axis movement was programmed after a contour transition element. **Error correction** - Edit the NC program.
1A0-0051	**Error message** Illegal axis programmed **Cause of error** You programmed an illegal axis. **Error correction** Edit the NC program: - Program another axis - Change the machining mode via FUNCTION MODE MILL/ TURN - Switch to another tool (with the appropriate tool type and tool orientation) - In the Programming mode, switch the AUTO DRAW soft key to ON and edit/check the (sub-) contour in a separate NC program - Edit/check the (sub-) contour in a separate NC program if the error with the AUTO DRAW soft key appears on ON
1A0-0052	**Error message** System error in the geometry chain: %1 **Cause of error** System error: Not all axis values were included in the message GmAxesValueSet. **Error correction** - Inform your service agency.
1A0-0053	**Error message** Programmed NC block not permitted right after a contour transition element **Cause of error** - Immediately after a transition element an attempt was made to program an NC block that is not allowed at that **Error correction** - Edit the NC program. - Inform your service agency.

Error number	Description
1A0-0054	**Error message** System error in the geometry chain: %1 **Cause of error** System error: The axis programmed in Cycle 19 is not a physical axis. **Error correction** - Inform your service agency.
1A0-0055	**Error message** System error in the geometry chain: %1 **Cause of error** System error: The axis programmed in Cycle 19 is a rotary axis. **Error correction** - Inform your service agency.
1A0-0056	**Error message** System error in the geometry chain: %1 **Cause of error** System error: The value in the message GmGeoRotWork-Plane is not absolute. **Error correction** - Inform your service agency.
1A0-0057	**Error message** Probe monitoring must not be switched on right after a contour transition element **Cause of error** The probe monitoring was switched on immediately after a contour element was programmed. **Error correction** - Complete the contour before switching on the touch probe.
1A0-0058	**Error message** TCPM must not be switched on or off right after a contour transition element **Cause of error** An attempt was made to switch the TCPM mode on or off immediately after a contour transition element. **Error correction** - Edit the NC program.

Error number	Description
1A0-0059	**Error message** %1 not allowed **Cause of error** A function was programmed that is not allowed in the current context. **Error correction** - Edit the NC program.
1A0-005A	**Error message** System error in the geometry chain: %1 **Cause of error** System error in the geometry chain. **Error correction** - Inform your service agency.
1A0-005B	**Error message** System error in the geometry chain: %1 **Cause of error** System error in the geometry chain. **Error correction** - Inform your service agency.
1A0-005C	**Error message** System error in the geometry chain: %1 %2 **Cause of error** System error in internal path calculation **Error correction** - Inform your service agency
1A0-005D	**Error message** Thread cycle not allowed right after a contour transition element **Cause of error** A thread cycle was programmed immediately after a contour transition element. **Error correction** - Edit the NC program.

Error number	Description
1A0-005E	**Error message** Tool not perpendicular to working plane **Cause of error** You tried to use a function that requires the tool to be perpendicular to the working plane (e.g. tapping) when it was not. **Error correction** - Edit the NC program - Rotate the tool to make it perpendicular to the working plane - If need be, tilt back the working plane
1A0-005F	**Error message** Thread cutting not allowed with tool compensation active **Cause of error** A tapping cycle was programmed with the tool compensation switched on. **Error correction** - First switch off the tool compensation.
1A0-0060	**Error message** Plunging depth in tapping cycle too small **Cause of error** Excessively small plunging depth was programmed in a tapping cycle. **Error correction** - Select a larger value for plunging depth.
1A0-0061	**Error message** Plunging depth in tapping cycle has wrong sign **Cause of error** In a tapping cyle the plunging depth was programmed with negative sign. **Error correction** - Edit the NC program.
1A0-0062	**Error message** Value for retraction has wrong sign **Cause of error** You programmed the retraction value with the incorrect algebraic sign. **Error correction** - Enter the algebraic sign

Error number	Description
1A0-0063	**Error message** System error in the geometry chain: %1 **Cause of error** - System error in the geometry chain. **Error correction** - Inform your service agency.
1A0-0064	**Error message** Tool change not allowed with tool compensation active **Cause of error** You programmed a tool change while the tool radius compensation was active. **Error correction** - Switch off the tool radius compensation before a tool change
1A0-0065	**Error message** Tool compensation must not be switched on before the approach block **Cause of error** You activated the tool radius compensation before the approach block. **Error correction** Switch on the tool radius compensation in the approach block
1A0-0066	**Error message** System error in the geometry chain: %1 **Cause of error** - System error in the geometry chain **Error correction** - Inform your service agency.
1A0-0067	**Error message** Departure movement not permitted immediately after activation of tool radius compensation **Cause of error** A departure was programmed immediately after switch-on of the tool compensation. **Error correction** Program a geometric element before departure.

Error number	Description
1A0-0068	**Error message** System error in the geometry chain: %1 **Cause of error** - System error in the geometry chain. **Error correction** - Inform your service agency.
1A0-006A	**Error message** System error in the geometry chain: %1 **Cause of error** - Sytem error in the geometry chain. **Error correction** - Inform your service agency.
1A0-006B	**Error message** System error in the geometry chain: %1 %2 **Cause of error** - System error in the geometry chain. **Error correction** - Inform your service agency.
1A0-006C	**Error message** System error in the geometry chain: %1 **Cause of error** - System error in the geometry chain. **Error correction** - Inform your service agency.
1A0-006D	**Error message** Approach movement not allowed before touch probe cycle **Cause of error** An approach was programmed before a probing cycle. **Error correction** - Edit the NC program.
1A0-006E	**Error message** System error in the geometry chain: %1 **Cause of error** System error in the geometry chain. **Error correction** - Inform your service agency.

Error number	Description
1A0-006F	**Error message** DEP movement not allowed right after a nonplanar movement **Cause of error** Illegal movement before a departure (DEP). **Error correction** Edit the NC program
1A0-0070	**Error message** Mid-program startup not possible **Cause of error** The target position of the mid-program startup is within a thread or measuring cycle. **Error correction** - Choose a different target position for mid-program startup
1A0-0071	**Error message** APPR/DEP CT or APPR/DEP LN not allowed with inactive tool radius compensation **Cause of error** You programmed APPR/DEP CT or APPR/DEP LN with inactive tool radius compensation. **Error correction** Edit the NC program
1A0-0072	**Error message** Approach movement cannot be calculated **Cause of error** The given geometric data do not result in a defined approach. **Error correction** Program another approach.
1A0-0073	**Error message** Departure movement cannot be calculated **Cause of error** The given geometric data do not result in a defined departure. **Error correction** Program another departure

Error number	Description
1A0-0074	**Error message** Circle tangent cannot be calculated **Cause of error** A circle was programmed with radius 0. **Error correction** Edit the NC program
1A0-0075	**Error message** After tangential departure only a line block is permitted **Cause of error** An illegal movement was programmed after a tangential departure. **Error correction** Program an L block after a tangential departure.
1A0-0076	**Error message** Rounding or chamfer cannot be calculated **Cause of error** Connecting geometrical element is too small or has the same tangent. **Error correction** Edit the NC program.
1A0-0077	**Error message** System error in the geometry chain: %1 **Cause of error** Cause is stated in the error text. **Error correction** Inform your service agency.
1A0-0078	**Error message** Illegal probe function **Cause of error** You tried to conduct a probe function during active mirroring or scaling. **Error correction** Edit the NC program

Error number	Description
1A0-0079	**Error message** No contour pockets present for fixed cycle **Cause of error** - No contour pocket geometery, or only islands defined **Error correction** - Program the missing Cycle 14 - If required, check the rotational direction and compensation direction of the individual contours
1A0-007A	**Error message** Invalid contour expression **Cause of error** Syntax error in contour expression: Missing parentheses, incorrect operands or operators, etc. **Error correction** - Check the contour expression
1A0-007B	**Error message** No technological data available for contour pocket machin-ing No technological data available for contour pocket machin-ing **Cause of error** A Cycle 20 must be programmed before every fixed cycle 21, 22, 23, or 24. **Error correction** Program a Cycle 20
1A0-007C	**Error message** Compensation spacing of tool paths too small **Cause of error** The compensation distance of the tool center path during contour cycle machining is less than 0.1 mm. **Error correction** – Use a larger tool radius, or - Increase the path overlap (Q2) (if relevant for the called cycle).
1A0-007E	**Error message** Polar programming not possible if M91/M92 is active **Cause of error** You tried to program polar coordinates during an active M91/M92. **Error correction** Program the positions with Cartesian coordinates

Error number	Description
1A0-007F	**Error message** Pole invalid **Cause of error** The pole was not programmed in the currently valid coordinate system. The error occurs when M91/M92 is used to define or apply the pole. **Error correction** Program a new pole
1A0-0080	**Error message** Arc definition in parallel coordinate axes **Cause of error** You programmed the arc end point in parallel coordinate axes (e.g. X and U coordinate). **Error correction** Program the arc end point in two coordinate axes that define a plane (e.g. X and V coordinate).
1A0-0081	**Error message** Invalid helix definition **Cause of error** You programmed a helical path with more than one coordinate outside the plane of the arc. **Error correction** Program a helix with one coordinate perpendicular to the plane of the arc.
1A0-0082	**Error message** Circle incorrectly programmed **Cause of error** Start point and end point are the same in a programmed arc. **Error correction** Program the start point and end point with different coordinates.
1A0-0083	**Error message** Circle incorrectly programmed **Cause of error** Radius for circular arc programming too small (radius of 0 or insufficient distance between start point and end point). **Error correction** - Increase the radius - Program the start point and end point with different coordinates.

Error number	Description
1A0-0084	**Error message** Circle incorrectly programmed **Cause of error** Center or end point of CC data is incorrect. The distances from the start point to the midpoint and from the end point to the midpoint differ by more than the tolerance value. **Error correction** The starting point, end point, and midpoint do not belong to a circular segment. - Recalculate the midpoint and/or end point. - If necessary, use another type of circular arc programming.
1A0-0085	**Error message** Circle incorrectly programmed **Cause of error** The end point of a CT block lies in the extension of the tangent of the previously programmed contour element. **Error correction** Edit the coordinates of the end point
1A0-0086	**Error message** Scaling factor invalid **Cause of error** - The programmed scaling factor is outside of the permitted range. **Error correction** - Enter a scaling factor in the range of 0.000 001 to 99.999 999.
1A0-0087	**Error message** Mid-program startup not allowed **Cause of error** Mid-program startup not allowed after an approach block that was programmed immediately before the program end. **Error correction** Set the mid-program startup to the block in which the approach movement is programmed.
1A0-0088	**Error message** Working plane cannot be changed **Cause of error** You called Cycle 10 (ISO: G73) before selecting the plane. **Error correction** Edit the NC program

Error number	Description
1A0-0089	**Error message** Illegal base transformation **Cause of error** The base information applies to a nonexistent axis. **Error correction** Change the datum.
1A0-008A	**Error message** Illegal coordinate transformation **Cause of error** The coordinate transformation applies to a nonexistent axis. **Error correction** Edit the NC program.
1A0-008B	**Error message** Incorrect axis index for FN18 **Cause of error** Coordinate transformations can be used only for the principal axes X,Y,Z. **Error correction** Edit the NC program.
1A0-008C	**Error message** Illegal coordinate transformation **Cause of error** Coordinate transformations can be used only for the principal axes X,Y,Z. **Error correction** Edit the NC program.
1A0-008D	**Error message** Not enough block memory **Cause of error** The system cannot resolve the NC program due to lack of block memory. Between two tool movements only a limited number of blocks can be processed that cause no movement (e.g.comments or variable assignments). **Error correction** Edit the NC program

Error number	Description
1A0-008E	**Error message** Safety clearance for synchronization of spindles is missing **Cause of error** No safety clearance was entered for synchronizing the spindle. **Error correction** Edit the NC program.
1A0-008F	**Error message** Tool radius too large **Cause of error** The radius of the cutter is too large to machine the programmed contour pockets or contour trains. **Error correction** Use a smaller tool radius or change the geometry of the contour pockets or contour trains.
1A0-0090	**Error message** Drill radius too large **Cause of error** - The pilot drill being used causes a collision with an edge contour. **Error correction** - Use a smaller drill or larger cutter. If the drill radius is equal to or less than the cutter radius, there can be no collisions.
1A0-0091	**Error message** R0 not allowed in contour subprogram. **Cause of error** Within a contour subprogram, RL/RR controls the type of contour (pocket/island). R0 is undefined and illegal. **Error correction** Remove R0 from the contour subprogram.
1A0-0092	**Error message** Rounding radius for inside corners too large **Cause of error** The circular arcs with the inside corner rounding radius programmed in Cycle 20 are too large to be inserted between two neighboring contour elements. **Error correction** Select a smaller rounding radius in Cycle 20.

Error number	Description
1A0-0093	**Error message** No space for approach movement **Cause of error** No suitable position was found for a collision-free approach in a contour pocket cycle (stroke of reciprocation during clearance, approach arc during finishing). **Error correction** Use a smaller tool diameter, change the pocket geometry, try drilling instead of a reciprocating plunge.
1A0-0094	**Error message** Cannot calculate tool radius compensation **Cause of error** Cannot calculate a tool radius compensation for the programmed contour in the working plane. **Error correction** Circles can be corrected only in the working plane.
1A0-0095	**Error message** Cannot calculate tool radius compensation **Cause of error** Cannot make a tool radius compensation for the programmed contour. **Error correction** Edit the contour or use another tool.
1A0-0096	**Error message** Contour subprogram: Illegal axis programmed **Cause of error** You programmed an illegal axis (e.g. a rotary axis) within a contour subprogram. **Error correction** Define in the contour subprogram only coordinates of the active working plane. Edit the NC program.

Error number	Description
1A0-0097	**Error message** Clearance height lies on machining side **Cause of error** The clearance height (Q7) programmed for the called cycle lies on the machining side of the workpiece surface (Q5) specified by the algebraic sign of the milling depth (Q1). That is not allowed. Also, the (absolute) clearance height (Q7) should not be equal to the (absolute) coordinate of the workpiece surface (Q5). **Error correction** - Change the cycle parameters clearance height (Q7), milling depth (Q1) and/or the coordinate of the workpiece surface (Q5).
1A0-0098	**Error message** Limit plane: Jump/jump not allowed **Cause of error** The "jump" method of machining cannot be set for both sides of a limit plane. **Error correction** Select another machining method for the limit plane.
1A0-0099	**Error message** Limit plane: jump move in wrong direction **Cause of error** A jump auxiliary movement must go toward the jump side of the limit plane. **Error correction** Change the direction of the jump auxiliary movement or adjust the clearance height
1A0-009A	**Error message** Limit plane: Incomplete definition of limit plane **Cause of error** Cannot acivate the plane because it has not been defined completely. A point or normal vector of the plane is missing, or maybe the direction for projection or for the jump auxiliary movements. **Error correction** Program the missing information before the plane is activat-ed.

Error number	Description
1A0-009B	**Error message** An active limit plane cannot be redefined or deleted **Cause of error** The definition of the limit plane cannot be changed or deleted as long as the limit plane is active. **Error correction** First deactivate the limit plane.
1A0-009C	**Error message** Cannot activate/deactivate limit plane now **Cause of error** The limit plane can be switched on or off only if the programmed position is on a curve side or a jump side beyond the 2nd safety clearance. **Error correction** Move to a safe position before switching the limit plane on or off.
1A0-009D	**Error message** Limit plane: Space between 2 limit planes must be set to "curve" **Cause of error** Between two limit planes (for non-parallel planes: in the work space) the curve machining method must be set. **Error correction** Adjust the machining method for one or both limit planes.
1A0-009E	**Error message** SL cycle call with active tool radius compensation **Cause of error** Before an SL cycle is called the tool radius compensation must be switched off. **Error correction** Program an R0 before the SL cycle or move the cycle call to another place.
1A0-009F	**Error message** An active zoning filter cannot be redefined or deleted **Cause of error** The definition of the zoning filter cannot be deleted or edited as long as the zoning filter is still active. **Error correction** First define the zoning filter.

Error number	Description
1A0-00A0	**Error message** Definition of the zoning filter incomplete **Cause of error** The zoning filter could not be activated because it was not fully defined. It still needs: at least one polygon (curve), a projection direction (of the length >0, if explicitly programmed), a setup clearance and a clearance height. **Error correction** Program the missing data before the zoning filter is activated.
1A0-00A1	**Error message** Definition of the zoning polygons (polygon curves) inconsistent **Cause of error** A closed zoning polygon must be defined by at least three points, an open polygon curve by at least two points. Each side of a polygon (curve) must have a component perpendicular to the direction of zoning projection. The individual polygon curves must not intersect with themselves or with the other polygon curves. An "illegal" zone must not enclose a "legal" zone. **Error correction** Check the definition of the zoning polygons (curves) and correct them.
1A0-00A2	**Error message** Incorrect sequence of vertices of a zoning polygon (polyg. curve) **Cause of error** The corner points of a polygon curve must be produced in the sequence: FirstPoint ->IntermediatePoint -> ... -> IntermediatePoint -> LastPoint(ForClose). The corresponding attributes must have the correct syntax. **Error correction** Note the correct sequence in the definition of zoning polygons.
1A0-00A3	**Error message** Invalid tool technology data **Cause of error** Technology data of the tool used, such as ANGLE or LCUTS, are not registered in the tool database or have invalid values. **Error correction** Make or correct the corresponding entries: ANGLE : Max. plunge angle in degrees, 1.0 <= ANGLE <= 90.0 LCUTS : Tooth length in mm, 0.1 <= LCUTS <= tool length

Error number	Description
1A0-00A4	**Error message** Tooth length of the tool used is too small **Cause of error** The tooth length of the tool used is smaller than the plunging depth programmed in the cycle. **Error correction** Program a smaller plunging depth or use a tool with a longer tooth.
1A0-00A5	**Error message** No contours are available under this label **Cause of error** Possible causes: - Contour labels that are be used as operands in expressions must have been assigned contours by means of DECLARE CONTOUR or by using previous expressions. - The contour can be empty, e.g. because antiparallel contour elements cancel each other. **Error correction** Check the contour declarations and expressions.
1A0-00A6	**Error message** Tool form compensation not allowed with active tool radius comp. **Cause of error** Tool form compensation (LN blocks with programmed workpiece normal) is not allowed during active tool radius compensation (RR/RL). **Error correction** Edit the NC program (e.g. R0 before first LN block).
1A0-00A7	**Error message** No machining side defined for peripheral milling **Cause of error** Before activation of the 3-D tool radius compensation (peripheral milling) no machining side was programmed with RR/RL. **Error correction** Change the NC program (e.g. RL or RR before the first LN or L block).
1A0-00A8	**Error message** Axis-specific scaling factor not allowed **Cause of error** During active tool radius compensation, axis-specific scaling factors are not allowed for circular and helical movements. **Error correction** Edit the NC program.

Error number	Description
1A0-00A9	**Error message** Operation not allowed with active radius compensation **Cause of error** A function was programmed (e.g. cycle, tool change) that is not allowed during active tool radius compensation. **Error correction** First switch off the tool radius compensation.
1A0-00AA	**Error message** Incorrect tool at start of block scan (current T%1, programmed T%2) **Cause of error** Mid-program startup with incorrect tool. **Error correction** Change to the correct tool and start again.
1A0-00AB	**Error message** Block scan cannot jump over probing functions **Cause of error** A probing function was found before the block being searched for with the block scan. **Error correction** Attempt the block scan on a different block
1A0-00AC	**Error message** Gap in non-cylindrical contour **Cause of error** The starting and end point of a non-cylindrical contour do not coincide. **Error correction** Edit the part program
1A0-00AD	**Error message** Contour intersects itself **Cause of error** The edge contour of a pocket cannot be programmed to intersect itself. **Error correction** Change the contour definition in the NC program.

Error number	Description
1A0-00AE	**Error message** Active tool larger than reference tool **Cause of error** To fully prevent collisions with the workpiece, the tool selected for machining with 3-D tool compensation must in every direction have a surface curvature that is not smaller than that of the reference tool. **Error correction** - Use a smaller tool. - Ensure that there are no collisions with the currently used tool, and use M107 to suppress this error.
1A0-00AF	**Error message** Axis-specific scaling not allowed **Cause of error** An axis-specific scaling factor is not allowed during the definition of a limit plane filter or zoning filter, or during active 3-D tool radius compensation or tool form compensation. **Error correction** Edit the NC program.
1A0-00B0	**Error message** Axis-value programming not allowed **Cause of error** Axis value programming in a linear block with 3-D tool radius compensation or tool form compensation is allowed only for rotary axes, not for translation axes. **Error correction** Use coordinate programming (remove M91)
1A0-00B1	**Error message** Rotary axes mixed with tool polar vectors **Cause of error** Rotary axis values and tool direction vectors must never be in the same NC block, and during active 3-D tool radius compensation they also must not appear alternately in successive blocks. **Error correction** - Program either only rotary axes or only one direction vector (TX, TY, TZ) to specify the tool axis direction per NC block. - During active 3-D tool radius compensation, program either only direction vectors or only rotary axes. If necessary, switch the tool radius compensation off and then on again within the program.

Error number	Description
1A0-00B2	**Error message** Programmed direction vector is zero vector **Cause of error** A programmed direction vector in an LN block must have at least one component not equal to zero. **Error correction** - Program NX, NY, NZ, TX, TY or TZ to be unequal to zero. - Remove NX, NY and NZ from the LN block (in which case there will be no tool form compensation!) or remove TX, TY and TZ (in which case the tool axis direction will not be changed).
1A0-00B3	**Error message** No spindle available for tapping **Cause of error** No spindle available for tapping **Error correction** Use a configuration with a spindle
1A0-00B4	**Error message** Missing start angle for incremental programming **Cause of error** If the end point of the last block lies on the pole, incremental angle programming is not allowed. **Error correction** Program the angle here with absolute values.
1A0-00B5	**Error message** Position logic not allowed in App/Dep movement **Cause of error** It is not allowed to switch on the "Positioning logic" during APPR/DEP movements. **Error correction** Switch off the "Positioning logic" before the APPR/DEP movement.
1A0-00B6	**Error message** Position logic not allowed in rounding/chamfer **Cause of error** It is not allowed to switch on the "Positioning logic" during rounding/chamfer movements **Error correction** Switch off the "Positioning logic" before the rounding/chamfer movement

Error number	Description
1A0-00B7	**Error message** Specified axis does not lie on "ToolSide"! **Cause of error** The specified axis does not lie on the "tool side" of the kinematic elements. **Error correction** The specified axis lies on the "tool side" of the kinematic elements.
1A0-00B8	**Error message** Specified axis does not lie on "WpSide"! **Cause of error** The specified axis does not lie on the "workpiece side" of the kinematic elements. **Error correction** Set the specified axis on the "workpiece side" of the kinematic elements.
1A0-00B9	**Error message** Specified axes inconsistent! **Cause of error** The specified axes are faulty **Error correction** Inform your service agency
1A0-00BA	**Error message** Programming of axes in the REF system is not allowed after RND/CHF or APPR/DEP **Cause of error** Programming of axes in the REF system is not allowed after RND/CHF or APPR/DEP movements **Error correction** Use RND/CHF or APPR/DEP movements without M91
1A0-00BB	**Error message** Programming together with M130 not allowed! **Cause of error** Programming together with M130 not allowed! **Error correction** This command cannot be used together with M130

Error number	Description
1A0-00BC	**Error message** Programming of axes in the REF system is not allowed **Cause of error** Polar programming is not allowed in the REF system! **Error correction** This command cannot be used together with M91
1A0-00BD	**Error message** Polar kinematics not possible! **Cause of error** Polar kinematics are not possible with the specified axes! **Error correction** The given axes cannot be used for polar kinematics
1A0-00BE	**Error message** General error in the calculation of a circle **Cause of error** No circle can be calculated from the given input data **Error correction**
1A0-00BF	**Error message** It is not possible to program a tangential connecting arc with axis-value programming. **Cause of error** - It is not possible to program a tangentially connecting arc during axis value programming **Error correction** - Program the arc with center and end point
1A0-00C0	**Error message** Pole and arc in various planes **Cause of error** The pole and end point of the arc are not programmed in the same plane **Error correction** Check the definitions of the pole and the arc
1A0-00C1	**Error message** Define the arc end point with two coordinates **Cause of error** Define the arc end point with two coordinates. **Error correction** Define the arc end point with both coordinates of the working plane.

Error number	Description
1A0-00C2	**Error message** Probe movement with length 0 **Cause of error** A probing movement must have a length not equal to zero **Error correction**
1A0-00C3	**Error message** Programming of rotary axes with circular movement is not allowed! **Cause of error** Programming of rotary axes with circular movement is not allowed! **Error correction**
1A0-00C4	**Error message** Departing movement not allowed! **Cause of error** Preceding movement is too complex **Error correction** Omit the departing movement or program the preceding movement differently
1A0-00C5	**Error message** System error in geometry chain: %1 **Cause of error** System error **Error correction** Inform your service agency
1A0-00C6	**Error message** Error in the geometry configuration. %1 **Cause of error** Indicated in English in the additional text **Error correction** Corresponding to the given cause
1A0-00C7	**Error message** Not allowed after approaching movement: %1 **Cause of error** Indicated in English in the additional text **Error correction** Corresponding to the given cause

Error number	Description
1A0-00C8	**Error message** Probing length must not be zero! **Cause of error** The probing length is zero. **Error correction** Correct the probing length (must be greater than 0)
1A0-00C9	**Error message** Nesting of pocket definitions is not allowed! **Cause of error** Nested pockets programmed **Error correction** Edit the program
1A0-00CA	**Error message** Deletion of the configuration object is not allowed! %1 **Cause of error** The type of deleted object is indicated in English in the additional text **Error correction** Use the configuration editor to reinsert the deleted object
1A0-00CB	**Error message** Contour train cannot be machined. **Cause of error** The contour description is ambiguous: The selected contour contains too many subcontours. **Error correction** Edit the NC program.
1A0-00CC	**Error message** Error in tool radius compensation **Cause of error** The tool radius compensation is missing or the entered tool radius compensation cannot be machined. **Error correction** Edit the NC program.
1A0-00CD	**Error message** Empty contour **Cause of error** An operand or intermediate result in the contour calculation is an empty contour. **Error correction** Edit the NC program.

Error number	Description
1A0-00CE	**Error message** A contour element of the slot in the cylinder surface too small **Cause of error** Possible causes: One distance is too short, or an angular length is too small. **Error correction** Edit the NC program.
1A0-00CF	**Error message** The data of the slot in the cylinder surface are faulty **Cause of error** Possible causes: A contour element of the slot is too small, the cylinder radius is too small, the slot too deep, or something similar. **Error correction** Edit the NC program.
1A0-00D0	**Error message** The position entered for the cylinder in the working plane is incorrect **Cause of error** Possible causes: Either the vectors do not have the length 1, they are not perpendicular, or there is some similar problem. **Error correction** Edit the NC program.
1A0-00D1	**Error message** Tool not perpendicular to the cylinder surface **Cause of error** The cylinder must be aligned parallel to the machine axes and be clamped at the center of the rotary table. The tool must be perpendicular to the cylinder surface. **Error correction** If needed, tilt the working plane in order to position the tool perpendicular to the cylinder surface, unless this has already been configured in the kinematics. - Program the position of the cylinder coordinate system correctly. - If required, configure the machining table system at the center over the rotary axis. The Z axis of the machine table system must point in the direction of rotary axis rotation.

Error number	Description
1A0-00D2	**Error message** No translation axis is parallel to the cylinder reference axis **Cause of error** Possible causes: The cylinder or the working plane is not in the right position. **Error correction** Edit the NC program.
1A0-00D3	**Error message** Approach not allowed **Cause of error** Possible causes: - APPR command in MDI mode - APPR command at the end of an NC program - APPR command before a coordinate transformation - Similar sequence of NC commands **Error correction** Edit the NC program.
1A0-00D4	**Error message** Reading the axis values is not allowed during block scan **Cause of error** During a block search you attempted to read the current axis values before finding the desired block. **Error correction** - Try a block scan to another block - Edit the program
1A0-00D5	**Error message** Contour subprogram for cylinder surface machining is faulty **Cause of error** Possible causes: - Programmed contour is not defined in X/Y cylinder surface coordinates. - Programmed contour contains incremental coordinates. - Programmed contour contains diameter coordinates. **Error correction** Edit the NC program.
1A0-00D6	**Error message** Contour subprogram for cylinder surface machining is faulty **Cause of error** Programmed contour is not defined in X/Y cylinder surface coordinates. **Error correction** Always program contours on a cylinder surface (regardless of the machine geometry) in X/Y cylinder surface coordinates.

Error number	Description
1A0-00D7	**Error message** No rotary axis found. **Cause of error** Possible causes: The first axis beneath the machine table must be a rollover rotary axis. **Error correction** - Change the kinematics configuration- Inform your service agency
1A0-00D8	**Error message** Cylinder axis is not parallel to rotary axis. **Cause of error** Possible causes: - The cylinder axis does not run parallel to the first axis under the machine table. The first axis under the machine table must be a rotary axis. - The cylinder is not clamped in the center. - One of the coordinate directions X, Y, or Z of the machine table system must point in the direction of the rotary axis. **Error correction** Change the kinematic configuration.
1A0-00D9	**Error message** Basic rotation not allowed **Cause of error** The rotary axis of the basic rotation during cylinder surface machining does not lie parallel to the cylinder axis. One of the coordinate directions X, Y, or Z of the machine table system must point in the direction of the rotary axis. **Error correction** - Change the basic rotation. - Change the kinematic configuration.
1A0-00DA	**Error message** Cylinder radius too small. **Cause of error** Cylinder radius too small. **Error correction** Edit the NC program.
1A0-00DB	**Error message** Cylinder slot too deep or excessive plunging depth. **Cause of error** Cylinder slot too deep or excessive plunging depth. **Error correction** Edit the NC program.

Error number	Description
1A0-00DC	**Error message** Cylinder slot too shallow or insufficient plunging depth. **Cause of error** Cylinder slot too shallow or insufficient plunging depth. **Error correction** Edit the NC program.
1A0-00DD	**Error message** Cylinder slot too narrow. **Cause of error** Cylinder slot too narrow. **Error correction** Edit the NC program.
1A0-00DE	**Error message** The programmed safety clearance is too small. **Cause of error** The clearance height must be be at least as large as the cutter radius. **Error correction** Edit the NC program.
1A0-00DF	**Error message** The precision for the slot walls is too small or too large. **Cause of error** The precision for the slot walls is too small or too large. **Error correction** Edit the NC program.
1A0-00E0	**Error message** Diameter of selected milling cutter is too small. **Cause of error** Diameter of selected milling cutter is too small. **Error correction** Insert a different tool.

Error number	Description
1A0-00E1	**Error message** Transformation not allowed **Cause of error** Possible causes: - Define datum shifts only in X/Y cylinder surface coordinates. - Contour definition with angle data: program a datum shift only within the contour definition. - Rotations and scaling are allowed only with length dimensions and only cylinder surface coordinates. - Define mirroring only in cylinder surface coordinates. - You must not change presets, basic rotations or the tilt condition during cylinder surface machining. **Error correction** Edit the NC program.
1A0-00E2	**Error message** This action is not allowed **Cause of error** Possible causes: This action is not yet implemented for cylinder surface machining. There may be a positioning block that is not programmed in the cylinder surface. Polar or axis-value programming, APPR or DEP blocks, probing blocks, thread cutting, tool change, certain cycles, 3-D tool compensation or similar things are also possible. **Error correction** Edit the NC program.
1A0-00E3	**Error message** The tool is in the wrong position **Cause of error** The tool is in an incorrect or unexpected position, e.g. too deep in the workpiece. This error also occurs during cylinder surface machining if the tool point is located too close to the cylinder axis. **Error correction** Edit the NC program, reposition the tool, or inform your service agency.
1A0-00E4	**Error message** Contour too complex **Cause of error** A pocket contour consists of more than 10000 blocks. **Error correction** Adapt the NC program: Program a simpler contour.

Error number	Description
1A0-00E5	**Error message** M103 not allowed **Cause of error** A reduction of the infeed velocity in the negative tool axis direction is not possible. Possible causes are programmed, e.g. rotary axes, 3-D radius compensation or kinematic compensation movements. **Error correction** Edit the NC program.
1A0-00E6	**Error message** Orientation not possible **Cause of error** You tried to orient the spindle through the NC, but no spindle is configured. **Error correction** Use a configuration with a spindle
1A0-00E7	**Error message** Closed contour train not allowed **Cause of error** The contour train is closed or nearly closed. **Error correction** Use pocket machining cycles for closed contour trains.
1A0-00E8	**Error message** Diameter of selected cutter is too large **Cause of error** The selected cutter does not fit in the cylinder slot. **Error correction** Insert a different tool.
1A0-00EA	**Error message** Rounding arc and chamfer not possible with active stretch filter **Cause of error** Transition elements (rounding arcs and chamfers) are not possible when the stretch filter is active. **Error correction** Switch off the stretch filter in the configuration (CfgStretch-Filter).

Error number	Description
1A0-00EB	**Error message** Programmed parallel axis not available **Cause of error** In the FUNCTION PARAXCOMP or FUNCTION PARAXMODE functions, you programmed parallel axes that are not available in this kinematic model. **Error correction** - Use other machine kinematics - Edit the NC program
1A0-00EC	**Error message** Position of a linear axis is overdetermined **Cause of error** You programmed two end values for the same machine axis in one NC program block. Possible causes: - You programmed the axis as coordinate and, over the syntax element POS, also as axis value - You did not switch with FUNCTION PARAXMODE to machining in minor axes or did not list the doubly defined axis in that function - With an active polar kinematic configuration you additionally assigned a target value to an axis in the three-axis kinematic configuration **Error correction** Edit the NC program.
1A0-00ED	**Error message** Syntax element POS not allowed in this block **Cause of error** You programmed the syntax element POS at an illegal place. **Error correction** Edit the NC program.
1A0-00EE	**Error message** Parallel axes not allowed in this block **Cause of error** You tried to program parallel axes - in approach or departure movements - in circle center or pole definitions - in circular or helical movements - in LN blocks. **Error correction** Edit the NC program.

Error number	Description
1A0-00EF	**Error message** Polar kinematics not allowed with parallel axis machining **Cause of error** You tried to use FUNCTION PARAXMODE although a polar kinematic configuration is active. **Error correction** Switch off the machining method before you switch another on.
1A0-00F0	**Error message** Kinematic switch-over is not allowed **Cause of error** You tried to use FUNCTION PARAXCOMP to program axis compensation values for parallel axes, although the default setting is not active. **Error correction** - Edit the NC program - Switch the kinematics only in the default condition. The default condition is configured in the config object CfgAxe-sPropKin in the parameter parAxComp
1A0-00F1	**Error message** Kinematic switch-over is not allowed **Cause of error** You have tried to perform a kinematics switch-over although FUNCTION PARAXMODE is active. **Error correction** Deactivate FUNCTION PARAXMODE before kinematic switch-over.
1A0-00F2	**Error message** Kinematic switch-over is not allowed **Cause of error** You tried to switch the kinematics although the tilted plane function or kinematic compensation movements (e.g. M128, M144) are active. **Error correction** Before switching the kinematics, deactivate all functions that depend on them.

Error number	Description
1A0-00F3	**Error message** Kinematic switch-over is not allowed **Cause of error** You tried to switch the kinematics although a kinematic compensation in a calling program (e.g. M128, M144, FUNCTION PARAXCOMP, FUNCTION PARAXMODE) is active. **Error correction** Before switching the kinematics, return to the default condition in all programs.
1A0-00F4	**Error message** No further axes allowed in pole plane **Cause of error** In a line block with polar coordinates you programmed further axes that lie in the working plane defined by the pole. The end position is therefore overdetermined. **Error correction** Delete additionally defined axes from the line block with polar coordinates.
1A0-00F5	**Error message** Tilting the working plane is not allowed **Cause of error** You tried to tilt the working plane although this function is not allowed with the active kinematic configuration. **Error correction** - Use another machine kinematic configuration - If required, edit the NC program - If necessary, inform your service agency
1A0-00F8	**Error message** Angle cannot be calculated **Cause of error** In the tilted-working-plane function you have the spatial-angle input mode active although this mode is not supported for your machine configuration. **Error correction** Select axis-angle input in the active kinematic table. If necessary, refer to your machine tool builder.

Error number	Description
1A0-00F9	**Error message** Coordinate transformation not allowed in the contour subprogram **Cause of error** You programmed an illegal coordinate transformation in a contour program, e.g. tilting the working plane, preset change, basic rotation or an axis offset. **Error correction** In the contour subprogram, use only the coordinate transformations rotation, datum shift, mirroring and scaling. Edit the NC program.
1A0-00FA	**Error message** Incremental tilting angle not allowed **Cause of error** You tried to tilt the working plane farther incrementally with axis angles, although you did not define the function with axis angles. **Error correction** Always define incremental tilting of the working plane with the same method as in the previous tilting. Change the type of incremental tilting or the previous absolute tilting.
1A0-00FB	**Error message** Incremental tilting angle not allowed **Cause of error** You tried to tilt the working plane farther incrementally with spatial angles, although you did not define the function with spatial angles. **Error correction** Always define incremental tilting of the working plane with the same method as in the previous tilting. Change the type of incremental tilting or the previous absolute tilting.
1A0-00FC	**Error message** No solution in limited range **Cause of error** You tried to limit the range for the master axis with SEQ+ or SEQ-. In this range the control cannot tilt the working plane. **Error correction** Cancel the limitation by SEQ+ / SEQ-.

Error number	Description
1A0-00FD	**Error message** Vectors are not perpendicular **Cause of error** You tried to define a working plane with PLANE VECTOR, entering vectors that are not perpendicular or do not intersect. **Error correction** Ensure that the vectors are perpendicular and intersect. Permit vectors that are not perpendicular to each other by changing the configuration: CfgRotWorkPlane-->autoCorrectVector to TRUE.
1A0-00FE	**Error message** Points on a plane too close to each other **Cause of error** You tried to define a working plane with PLANE POINTS, entering plane points that lie too close together. **Error correction** Define plane points that are farther apart from each other.
1A0-00FF	**Error message** Points on a plane lie on a line **Cause of error** You tried to define a working plane with PLANE POINTS, entering plane points that lie one a straight line. Therefore the control cannot calculate an unambiguous plane. **Error correction** Define plane points that form a triangle.
1A0-0100	**Error message** No rotary axes available **Cause of error** You attempted to tilt the working plane in a kinematic configuration without rotary axes. **Error correction** - Correct the NC program: Delete the functions for tilting the working plane. - Activate the kinematic configuration with rotary rotary axes.

Error number	Description
1A0-0101	**Error message** Vectors are too short **Cause of error** You tried to define a working plane with PLANE VECTOR, when at least one of the vectors is too short. **Error correction** - Enter longer vectors - Adapt machine parameters (must be done by machine manufacturer): Set machine parameter CfgRotWorkPlane/autoCorrectVector to the value TRUE in order to permit basic vectors with zero length.
1A0-0102	**Error message** Infeed depth too small **Cause of error** In one of the Cycles 21, 22, 24 or 25 (DIN/ISO: G121, G122, G124, G125) you defined the plunging depth Q10 smaller than 0.1 mm. **Error correction** Define the plunging depth Q10 with a value larger than 0.1 mm.
1A0-0103	**Error message** Too many rotary axes available **Cause of error** You attempted to tilt the working plane in a kinematic configuration with more than two rotary axes. This is only possible when programming with axis values. **Error correction** - Correct the NC program: use PLANE AXIAL - Activate the kinematic configuration with two rotary rotary axes - Use M138 to select the rotational axes
1A0-0104	**Error message** Combination of functions not permitted **Cause of error** You tried to tilt the working plane while a datum shift in rotational axes was active. This is only possible when programming with axis values. **Error correction** - Correct the NC program: use PLANE AXIAL - Use a reference point for this shift.

Error number	Description
1A0-0105	**Error message** Position after G43/G44 not perpendicular to the G41/G42 contour! **Cause of error** The G43/G44 approach is not perpendicular to the next G41/G42 movement. This causes a contour error that can damage the workpiece. The contour error depends on the distance between the end position and the perpendicular distance R of the G41/G42 starting position. The distance is greater than 0.1 * tool radius R. **Error correction** - Edit the NC program: the approach from G43/G44 must be perpendicular to the contour.
1A0-0106	**Error message** & operator not applicable to active 3-axis kinematics **Cause of error** You tried to use the X, Y or Z axis with the & operator, although this axis in included in the active 3-axis kinematic configuration. The & operator is permissible only if you have used the PARAXMODE FUNCTION to take the corresponding axis out of the active 3-axis kinematic configuration. **Error correction** Use the corresponding axis without the & operator.
1A0-0107	**Error message** M128 and M144 with CYCL CALL POS not allowed **Cause of error** During an active M128 or M144 you tried to use CYCL CALL POS to call a cycle. **Error correction** Deactivate M128 and M144 before the cycle call with CYCL CALL POS.
1A0-0108	**Error message** CYCL CALL POS: Incremental values without reference **Cause of error** You tried to call CYCL CALL POS with incremental coordinates, of which some are not based on coordinates that you had previously programmed with CYCL CALL POS. **Error correction** Ensure that every incrementally programmed coordinate in a cycle call with CYCL CALL POS is given with respect to coordinates in previously programmed CYCL CALL POS commands.

Error number	Description
1A0-0109	**Error message** Resumption with M120 not allowed **Cause of error** Re-entry with GOTO during active M120 not permitted. **Error correction** Re-entry possible only via mid-program startup.
1A0-010A	**Error message** TCPM: PATHCTRL VECTOR not possible **Cause of error** You tried to move the rotary axes with PATHCTRL VECTOR so that the direction of the tool axis is always in the same plane. PATHCTRL VECTOR is not possible with the programmed direction vectors. **Error correction** - Use the TCPM function with PATHCTRL AXIS. - If required, correct the directional vector.
1A0-010B	**Error message** Spatial circular arc not allowed **Cause of error** You tried to move on a spatial circular arc, but this function is not possible. A spatial circular arc results, for example, if you have programmed a circle in the X/Z plane, and then, for example, turn with Cycle 10 in the X/Y plane. **Error correction** Edit the NC program.
1A0-010C	**Error message** Movement is nonlinear **Cause of error** You programmed an NC block that causes a counter axis to make a nonlinear movement. **Error correction** - Edit the NC program
1A0-010D	**Error message** This contour is not possible in turning cycles. **Cause of error** You programmed an illegal NC block within a contour for a turning cycle. **Error correction** Edit the contour definition in the NC program.

Error number	Description
1A0-010E	**Error message** Run-in length incorrectly programmed **Cause of error** You programmed the overrun length for tapping to be zero or a negative number. **Error correction** Enter only positive numbers for the run-in length. Recommended run-in length: at least half the pitch.
1A0-010F	**Error message** Overrun length incorrectly programmed **Cause of error** You programmed the overrun length for tapping to be zero or a negative number. **Error correction** Enter only positive numbers for the overrun length. Recommended overrun length: at least half the pitch.
1A0-0110	**Error message** Error in turning cycle **Cause of error** Possible causes: - You tried to run a turning cycle, although the active tool is not a turning tool. - You tried to run a turning cycle, although the milling mode is active. **Error correction** - Insert a turning tool. - Use FUNCTION MODE TURN to switch to the turning mode.
1A0-0111	**Error message** Error in rounding arc **Cause of error** You tried to program a rounding arc in a contour with less than two elements. **Error correction** Edit the NC program.
1A0-0112	**Error message** Error in rounding arc **Cause of error** You tried to end a contour with a rounding arc. However, the contour not closed. **Error correction** Edit the NC program.

Error number	Description
1A0-0113	**Error message** Error in contour element of a turning cycle **Cause of error** You programmed an undercut or recess at the beginning of a turning contour. **Error correction** Edit the NC program.
1A0-0114	**Error message** Error in recess **Cause of error** Possible causes: - You have programmed neither the center point of the recess (CENTER) nor the recess position (PLACE). - You have programmed both the center point of the recess (CENTER) and the recess position (PLACE). **Error correction** Edit the NC program.
1A0-0115	**Error message** Turning operation: tool position is incorrect **Cause of error** The cutting insert of the turning tool is not in the permitted working plane. **Error correction** Rotary axes and cutting inserts must lie in a working plane. - Correct the position of the tool. - If desired, adopt the rotary axis coordinates with M128 or M144.
1A0-0116	**Error message** Recessing cycle: tool position is incorrect **Cause of error** You tried to use a recessing tool, although the machine is not in the default setting. **Error correction** Cancel the tool orientation: - Move the rotary axis to the home position. - If necessary, use M128 or M144 to load the rotary axis values.

Error number	Description
1A0-0117	**Error message** Turning cycle with button tool dimensioned at center **Cause of error** You tried to run a turning cycle with a button tool dimensioned at its center. **Error correction** Button tools that are dimensioned at the center must be measured in a corner, so they have to have tool orientation 1 to 8.
1A0-0118	**Error message** Program resumption not possible **Cause of error** It is not possible to resume the program at the point of interruption. **Error correction** - Jump to the program start using the GOTO key or reselect the program over PGM MGT. - The StretchFilter must be deactivated in order to be able to continue the program at the point of interruption.
1A0-0119	**Error message** Thread not possible with inclined tool **Cause of error** You tried to cut a thread with an inclined tool and at a feed rate of 0. **Error correction** - Cancel the tool position. To do so, return the rotary axes to their home position and, if required, take over the rotary axis values with M128 or M144. - Change the type of feed rate.
1A0-011A	**Error message** Coordinate transformation in turning operation **Cause of error** You tried to change to turning mode although a coordinate transformation (rotation, mirroring and/or scaling) is active. **Error correction** Deactivate coordinate transformations (rotation, mirroring and or scaling) before you switch to turning mode.

Error number	Description
1A0-011B	**Error message** Noncylindrical contour must not begin at an inside corner **Cause of error** You tried to begin a noncylindrical contour with radius compensation at an inside corner (concave position). These are some of the possible causes of this error message: - The starting point lies on a real inside corner - The starting point lies on the transition between two arcs that are tangential, but were programmed or generated with poor accuracy - A "round" inside noncylindrical contour was resolved into nothing but straight-line segments. The starting point then lies on a tangential transition between two arcs. However, the resolution into straight line segments again results in an inside corner. **Error correction** - Correct the starting point by choosing a correct point that does not lie on an inside corner - Create/generate the noncylindrical program (especially circular movements) with higher accuracy - Avoid resolving the noncylindrical contour into straight-line segments, or manually place the starting point in the middle of a generated line segment - Have the tool-compensated path of the noncylindrical contour calculated/generated externally instead of by the control.
1A0-011C	**Error message** Side finishing: Tool radius too large **Cause of error** With the defined contour data, the tool radius is too large for side finishing or fine roughing. **Error correction** For side finishing, the sum of the side-finishing allowance (side-finishing cycle) and the finishing-tool radius must be smaller than the sum of the side-finishing allowance (contour data cycle) and the roughing-tool radius. The above calculation also applies if you run the side-finishing cycle without first having cleared the area out with the roughing tool; the radius of the roughing tool then has the value “0.” During fine roughing, the radius of the fine roughing tool must be smaller than the radius of the coarse roughing tool.

Error number	Description
1A0-011D	**Error message** Normal vector too short **Cause of error** You tried to define a working plane with PLANE VECTOR, but the normal vector is too short. **Error correction** The TNC cannot correct the normal vector automatically, enter a longer normal vector.
1A0-011E	**Error message** Workpiece blank definition: contour is not closed **Cause of error** The start point and end point do not agree in the definition for the workpiece blank contour. **Error correction** Edit the NC program. The start position must be complete, which means that it has to contain values for both coordinates.
1A0-011F	**Error message** Invalid tooth length in recessing cycle **Cause of error** You tried to run a recess turning cycle with a tool whose tooth length was zero or is not defined. **Error correction** Check the cutting length for the recess turning tool in the tool table.
1A0-0120	**Error message** No recessing tool is active **Cause of error** You tried to run a recess turning cycle although the active tool is not a recess turning tool. **Error correction** Insert a recess turning tool.
1A0-0121	**Error message** Illegal syntax element in contour subprogram for turning cycles **Cause of error** In a contour subprogram you have not specified the radius compensation in the first line of the contour description. **Error correction** Correct the contour subprogram.

Error number	Description
1A0-0123	**Error message** Radius comp. undefined **Cause of error** You programmed a radius-compensated single-axis positioning block that, without the radius compensation, does not result in tool movement (e.g. IX+0 R+, ISO: G7). **Error correction** Edit the NC program.
1A0-0124	**Error message** Cannot calculate any intersection **Cause of error** The control cannot calculate an intersection: two straight lines are parallel. **Error correction** Check the contour description.
1A0-0125	**Error message** Cannot calculate any intersection **Cause of error** The control cannot calculate an intersection: two outside circles do not intersect. **Error correction** Check the contour description.
1A0-0126	**Error message** Cannot calculate any intersection **Cause of error** The control cannot calculate an intersection: two inside circles do not intersect. **Error correction** Check the contour description.
1A0-0127	**Error message** Cannot calculate any intersection **Cause of error** The control cannot calculate an intersection: two inside circles do not intersect. **Error correction** Check the contour description.

Error number	Description
1A0-0128	**Error message** Cannot calculate any intersection **Cause of error** The control cannot calculate an intersection: a straight line and a circle do not intersect. **Error correction** Check the contour description.
1A0-0129	**Error message** Contiguous circles with opposite direction of rotation **Cause of error** Two contiguous circles illegally have opposite directions of rotation. **Error correction** Check the contour description.
1A0-012A	**Error message** Contour element with length=0 programmed **Cause of error** The starting the ending points of a profile contour are identical. **Error correction** Check the contour description.
1A0-012B	**Error message** Cannot calculate any intersection **Cause of error** The control cannot calculate an intersection: a circle has a zero radius. **Error correction** Check the contour description.
1A0-012E	**Error message** Cannot calculate any intersection **Cause of error** The control cannot calculate an intersection: two straight lines are parallel and are not superimposed. **Error correction** Check the contour description.

Error number	Description
1A0-012F	**Error message** Faulty subdivision of the vertical profiles **Cause of error** The control cannot machine the defined profile. **Error correction** Check the profile description (SEL CONTOUR PROFILE), machining parameters and data of the tool used.
1A0-0130	**Error message** The plane profile is not closed or begins at a corner **Cause of error** The start point and end point of the plane profile either do not match or they are on a corner. **Error correction** Check the programmed plane profile.
1A0-0131	**Error message** Control cannot conclude machining operation **Cause of error** Error in internal calculations. **Error correction** Check the contour description (SEL CONTOUR SURFACE and SEL CONTOUR PROFILE), machining parameters and data of the tool used.
1A0-0132	**Error message** Machining operation requires too many cycles **Cause of error** The maximum permissible number of machining runs was exceeded. **Error correction** Check the machining parameters and data of the tool used.
1A0-0133	**Error message** Error management tables are inconsistent **Cause of error** SEL CONTOUR SURFACE uses a table index that lies outside of the permissible value range. **Error correction** Inform your service agency.

Error number	Description
1A0-0134	**Error message** Cycle not suited for contour definition with APPR/DEP block. **Cause of error** The cycle you selected cannot use any contour definition that contains APPR or DEP blocks. **Error correction** Remove APPR and DEP blocks from the contour definition. For Cycle 1025, use Cycle 270 to program APPR/DEP.
1A0-0135	**Error message** APPR block in contour definition not in first line **Cause of error** In a contour description you have programmed an APPR block that is not in the first line. **Error correction** Correct the contour description.
1A0-0136	**Error message** Too many APPR/DEP blocks in contour description **Cause of error** You programmed multiple APPR or DEP blocks in a contour description. **Error correction** Contour descriptions must contain only one APPR/DEP block. Correct the contour description.
1A0-0137	**Error message** Radius compensation in contour subprogram inconsistently defined **Cause of error** In a contour subprogram, you have defined multiple radius compensation values that contradict each other. **Error correction** Edit the contour subprogram.
1A0-0138	**Error message** DEP block in contour definition not programmed in the last line **Cause of error** In a contour subprogram you have programmed an DEP block that is not in the last line of the contour description. **Error correction** Correct the contour description.

Error number	Description
1A0-013A	**Error message** Erroneous plane profile programmed **Cause of error** The plane profile was programmed incorrectly. **Error correction** - Ensure that for each geometric element that describes the geometry of the plain profile (which therefore excludes blocks that describe an approaching or departing motion) an appropriate vertical profile is defined. - Ensure that in each case no more than one approaching or departing motion is programmed at the beginning or end of the plane profile definition. - Ensure that at first, after each reference to a vertical profile, at least one geometric element is programmed.
1A0-013B	**Error message** Illegal transformation during execution of a profile surface **Cause of error** Programming a profile surface (Cycles 281-283) is impermissible if a mirror image along a tool axis (Cycle 8) is active at the same time. **Error correction** Rewrite the program so that the mirroring is not active during the call of the profile surface cycles (281-283).
1A0-013C	**Error message** Cycles 281 and 283 only applicable to pockets **Cause of error** Cycles 281 for roughing and 283 for floor finishing must be used only for profile surfaces that form a pocket. **Error correction** Ensure that the programmed plane profile is closed. If the plane profile is defined with Cycle 14, ensure that the combination of machining direction, radius compensation and monotonicity of the vertical profile results in a pocket.
1A0-013D	**Error message** Missing radius compensation in the plane profile definition **Cause of error** If the plane profile of a profile surface (Cycles 280 to 283) is programmed using Cycle 14, a radius compensation (RL or RR) must be entered within the definition of the plane profile. **Error correction** Enter the radius compensation within the definition of the plane profile.

Error number	Description
1A0-013E	**Error message** Invalid vertical profile defined **Cause of error** The programmed vertical profile of a profile surface(Cycles 280 – 283) is invalid. **Error correction** Ensure that the definition of the vertical profile contains at least two geometric blocks. Ensure that no radius compensation is programmed within the definition of the vertical profile. Ensure that the vertical profile increases with respect to the abscissa (usually the x coordinate) (i.e. increases monotonically). Ensure that the vertical profile either increases or decreases with respect to the ordinate (usually the y coordinate) (i.e. monotonically increases or monotonically decreases).
1A0-013F	**Error message** Radius compensation defined within plane profile **Cause of error** If the plane profile of a profile surface (Cycles 280 to 283) is specified using the syntax element CONTOUR DEF, there must be no radius compensation entered within the definition of the plane profile. **Error correction** Remove the radius compensation from the definition of the plane profile.
1A0-0140	**Error message** Profile surface cycle with active tool radius compensation **Cause of error** You tried to call a profile surface cycle (Cycles 281 – 283) with active tool radius compensation. **Error correction** Before the profile surface cycle, use R0 to cancel the tool radius compensation or program the cycle call at another location.
1A0-0141	**Error message** Programmed plane profile is not closed. **Cause of error** You tried to use CONTOUR DEF to program a plane profile as a pocket. However, the plane profile does not result in a closed contour. **Error correction** Program a closed plane profile. Use Cycle 14 to define an open plane profile.

Error number	Description
1A0-0142	**Error message** Programmable axis %1 missing in "CfgChannelAxes/progAxis" **Cause of error** The machine configuration is faulty. The affected axis is not configured as a programmable axis. **Error correction** - Inform your machine tool builder - Correct the machine configuration: Enter the axis as a programmable axis in 'CfgChannelAxes/progAxis'
1A0-0143	**Error message** Roughing cycle: Start position too close to rotary axis **Cause of error** The starting position of a roughing cycle is located too close to the rotary axis. **Error correction** Edit the NC program.
1A0-0144	**Error message** Workpiece blank definition: Contour intersects itself **Cause of error** A contour that describes the surface line of a workpiece blank (NC block BLK FORM ROTATION) intersects itself. **Error correction** Correct the NC program: Adapt the workpiece blank contour so that it no longer intersects itself.
1A0-0145	**Error message** Workpiece blank definition: Illegal axis in subprogram **Cause of error** You programmed an illegal axis in a subprogram that defines the surface line of a workpiece blank (NC block: BLK FORM ROTATION). **Error correction** - Correct the NC program: Program the coordinates in the subprogram that result from the selection of the rotation axis for the workpiece blank.
1A0-0146	**Error message** Turning tool with contradictory data **Cause of error** A turning tool was inserted whose data are contradictory for the following reasons: The tool orientation does not fit the tool angle and point angle. **Error correction** Correct the data in the turning tool table

Error number	Description
1A0-0147	**Error message** Workpiece blank definition for active coordinate transformation **Cause of error** During an active coordinate transformation (datum shift, tilting), you tried to define a workpiece blank using a surface line (NC block: BLK FORM ROTATION). **Error correction** Reset all active coordinate transformations before you define the workpiece blank.
1A0-0148	**Error message** Function not permitted **Cause of error** You tried to calculate a spindle rotation between input coordinate system and tool coordinate system (e.g. with ID210 NR8), at the same time as a transformation for scaling or mirroring was active. **Error correction** Edit the NC program
1A0-014B	**Error message** Illegal movement if FUNCTION TCPM and mirroring are active. **Cause of error** You tried to program a rotary axis motion defined with spatial angles during a circular movement (CP..., CPT...) with active TCPM FUNCTION (with AXIS SPAT) and active mirroring. **Error correction** Deactivate mirroring before the motion is processed while TCPM FUNCTION is active.
1A0-014C	**Error message** Erroneous tool-carrier kinematics in the file %1 **Cause of error** - Tool-carrier kinematic model in the given file is faulty **Error correction** - For more detailed information on this error message, press the INTERNAL INFO soft key - Check the tool kinematic model in the given file and correct it if necessary - Inform your service agency

Error number	Description
1A0-014D	**Error message** Data of the tool carrier kinematics is faulty **Cause of error** The tool-carrier kinematic data for the current tool are faulty. Note: Do not run any NC programs in this condition and be careful if you move the axes manually! **Error correction** - Ensure under “KINEMATICS” that a valid file for the tool-carrier kinematics is entered for the desired tool in the tool table. - Acknowledge the error message - Run a TOOL CALL for a tool that has no tool-carrier kinematics assigned or one that has a valid tool-carrier kinematic model. - Inform your service agency.
1A0-0151	**Error message** Illegal number of lines (%1) in the compensation value table **Cause of error** Too few (or too many) measured values were entered in the compensation value table for 3D-ToolComp. At least two values must be entered. **Error correction** - Check the compensation value table and correct it if necessary - Perform the calibration cycle again
1A0-0152	**Error message** Inconsistent (multiple) angular values in the compensation table. **Cause of error** The compensation value table contains ambiguous (multiple) angular values (ANGLE). **Error correction** Check the table and perform the calibration cycle again if necessary.
1A0-0153	**Error message** Angle value (%1) is outside of the valid range. **Cause of error** An evaluation was attempted outside the boundaries of the angular range that was measured. **Error correction** Expand the compensation table in order to include the necessary angles.

Error number	Description
1A0-0154	**Error message** Tool carrier kinematics is ignored in "%1" **Cause of error** The given kinematic model has no insertion point for tool-carrier kinematics. **Error correction** - Adjust the machine configuration for tool-carrier kinematics. To do so, please contact your machine tool builder. Please note: If you use the entered kinematic model, the machining operation will be executed without the desired tool carrier.
1A0-0161	**Error message** Programmed rotary axis on circle not allowed (TCPM TIP-CENTER) **Cause of error** An attempt was made to program a circle with a rotary axis setting without tool radius compensation with an active TCPM REFPNT TIP CENTER. Simultaneous rotary axis adjustment on circles is permissible only with tool radius compensation. **Error correction** Activate tool radius compensation
1A0-0162	**Error message** Tool radius compensation incorrectly ended (TCPM TIP-CENTER) **Cause of error** An attempt was made to stop tool radius compensation with TCPM REFPNT TIP-CENTER in an impermissible manner. With this TCPM preset, tool radius compensation can be stopped only by means of a linear block with R0 in which both working plane coordinates are programmed. **Error correction** Stop tool radius compensation with a linear block containing both working plane coordinates.
1A0-0164	**Error message** Cannot find direction of rotation for the defined contour **Cause of error** The programmed workpiece blank contour is not closed or is without curves. It is therefore impossible to find the rotational direction or display in 3-D. **Error correction** - Change the NC program - Program a workpiece blank contour consisting of more than one point. The contour must be closed and must not lie on a single straight line.

Error number	Description
1A0-0165	**Error message** Recesses in the BLK FORM are not supported **Cause of error** A recess was programmed in the contour for the BLK FORM. **Error correction** - Edit the NC program. Remove the recess from the contour for the BLK FORM.
1A0-0166	**Error message** Undercuts in the BLK FORM are not supported **Cause of error** An undercut was programmed in the contour for the BLK FORM. **Error correction** Remove the undercut from the contour
1A0-0167	**Error message** SL cycle not allowed (TCPM REFPNT CENTER) **Cause of error** An attempt has been made to program an SL cycle while TCPM REFPNT TIP-CENTER or REFPNT CENTER-CENTER is active. No SL cycles are possible with these TCPM settings. **Error correction** Edit the NC program
1A0-0169	**Error message** M130 not allowed **Cause of error** No M130 can be programmed during active turning-tool compensation in the tilted plane's system (FUNCTION TURNDATA CORR-WPL, or columns WPL-DZL and WPL-DX-DIAM from the turning-tool table). **Error correction** - Check the NC program and adapt it if necessary.
1A0-016C	**Error message** Facing slide: Only ZK contour allowed **Cause of error** A contour was programmed that is not in the ZX plane. **Error correction** Check the NC program and adapt it if necessary.

Error number	Description
1A0-016D	**Error message** Facing slide: Transformation not allowed **Cause of error** Certain transformations are not allowed with an active facing slide: - Tilting the working plane - Scaling - Datum shift **Error correction** Check the NC program and adapt it if necessary.
1A0-016E	**Error message** Tilted working plane no allowed **Cause of error** With FUNCTION MODE TURN, tilting the working plane is allowed only with facing slide kinematics. **Error correction** Check the NC program and adapt it if necessary.
1A0-016F	**Error message** Helix with active facing slide not allowed **Cause of error** No helix is allowed with an active facing slide. **Error correction** Check the NC program and adapt it if necessary.
1A0-0170	**Error message** Facing slide: Combination with M91 not allowed **Cause of error** The combination of active facing slide and M91 are not allowed. **Error correction** Check the NC program and adapt it if necessary.
1A0-0171	**Error message** Facing slide: TCPM not allowed **Cause of error** No TCPM (M128) is allowed with an active facing slide. **Error correction** Check the NC program and adapt it if necessary.

Error number	Description
1A0-0172	**Error message** Facing slide: 3-D radius compensation not allowed **Cause of error** 3-D radius compensation is not allowed with an active facing slide. **Error correction** Check the NC program and adapt it if necessary.
1A0-0173	**Error message** Special kinematics are not allowed for facing slide machining **Cause of error** You tried to combine FUNCTION PARAXMODE with a special kinematic model: - Polar kinematics - Facing slide kinematics **Error correction** Deactivate the machining method before another is to be activated.
1A0-0174	**Error message** Facing slide: Only in turning mode **Cause of error** A facing slide can be activated only in turning mode. **Error correction** Use FUNCTION MODE TURN to switch to the turning mode.
1A0-0175	**Error message** Facing slide: Impermissible circle programmed **Cause of error** The programmed circle has a radius or arc that is too small for the facing slide **Error correction** Check the NC program and adapt it if necessary
1A0-0176	**Error message** Facing slide: Probing block not allowed **Cause of error** Probing block not allowed with an active facing slide **Error correction** Run the probing block before the facing slide is activated

Error number	Description
1A0-0177	**Error message** Facing slide: Spindle is not aligned **Cause of error** The Z axis of the input system and the spindle direction are not parallel **Error correction** Align the spindle direction before the facing slide is activated
1A0-0178	**Error message** Facing slide not possible when stretch filter is active **Cause of error** Facing slide cannot be activated when the “stretch filter” is active **Error correction** - Check the entry under CfgStrechFilter and change it if required - Inform your service agency
1A0-0179	**Error message** Facing slide: M140 not allowed **Cause of error** Tool retraction (M140) not allowed with an active facing slide **Error correction** Edit the NC program
1A0-017A	**Error message** Facing slide: PARAXCOMP not allowed **Cause of error** FUNCTION PARAXCOMP not allowed with an active facing slide **Error correction** Check the NC program and adapt it if necessary
1A0-017B	**Error message** Turning cycle is allowed only with an active facing slide **Cause of error** With facing slide kinematics, you tried to execute a turning cycle without activating the facing slide **Error correction** - Program FACING HEAD POS

Error number	Description
1A0-017C	**Error message** Tool contour missing for simultaneous turning cycle **Cause of error** The tool contour for simultaneous turning could not be read. **Error correction** The 2-D tool contour for the cycle is calculated from the corresponding 3-D tool carrier kinematics: - Ensure that a valid tool carrier kinematic model is entered in the "KINEMATIC" tool table column. - For the simultaneous rotation cycle, make particularly sure that the geometry defined in it matches the data of the tool in the tool table.
1A0-017D	**Error message** Faulty tool data for simultaneous turning cycle **Cause of error** The tool data of the tool table is not compatible with the simultaneous turning cycle. The tool data (ZL, XL, RS, TO, P-ANGLE, T-ANGLE, CUTWIDTH, CUTLENGTH and KINEMATIC) must describe a realistic tool. In particular, the following conditions must be met: - Neither the radius (RS) nor the cutting edge length (CUTLENGTH and CUTWIDTH) are zero. - Only button tools, roughing tools, and finishing tools are allowed. - TO, ZL, and XL must agree with the tool holder geometry in KINEMATIC. **Error correction** Check and correct the entries in the tool table
1A0-017E	**Error message** GS rotation with working plane not in XY **Cause of error** An attempt has been made to combine a rotation from the global program settings with a working plane in ZX or YZ. This is not allowed. Such a working plane is available in the turning mode as well as when using TOOL CALL X or TOOL CALL Y. **Error correction** Edit the NC program or, in the corresponding program section, activate no rotation via global program settings.

Error number	Description
1A0-017F	**Error message** Negative cutting edge radius **Cause of error** An attempt was made to start a turning cycle with a turning tool with negative cutting edge radius. That is not allowed. The effective cutting edge radius is the sum of the following three elements: - Value in the RS column from the turning-tool table - Value in the DRS column from the turning-tool table - through FUNCTION TURNDATA CORR-TCS: Z/X DRS programmed oversize **Error correction** The sum of the three values must be positive: adjust the NC program or tool table
1A0-0180	**Error message** Simultaneous turning: No fitting tilting axis found. **Cause of error** No appropriate tilting axis was found for the simultaneous turning cycle. **Error correction** - If the machine physically has a suitable axis: - Adapt the precession angle via Cycle 800 - Check the kinematics configuration and change it if necessary - Inform your service agency
1A0-0182	**Error message** Tilting working plane does not fit to rotary transformations **Cause of error** - An attempt has been made to activate the tilting of the machining plane while one of the rotational transformations mentioned below is active. - An attempt has been made to activate one of the rotational transformations named below while the machining plane is tilted. **Error correction** - Deactivate tilting of the machining plane or Cycle 800. If the transformation named in item 2 is active outside the turning operation, please contact the machine manufacturer. The following are affected: 1. Transformation activated by Cycle 800 2. Special transformation in turning operation, which the input system usually adapts to kinematics with the A or B table. The POS tab of the status display shows whether the transformer is active.

Error number	Description
1A0-0183	**Error message** Data for turning tool invalid **Cause of error** The data of the active turning tool are invalid. Invalid value combination: TYPE and TO do not match. The value 9 can be defined in TO only for roughing and finishing tools. **Error correction** Edit the tool data.
1A0-0184	**Error message** Simultaneous turning: Programmed tool compensations not allowed **Cause of error** Programmed tool compensations (FUNCTION TURNDATA CORR-TCS ...) are not permitted for the simultaneous turning cycle. Such compensations change the position of the indexable insert relative to the tool carrier, which can lead to collisions. **Error correction** Remove all tool compensations programmed before the cycle.
1A0-0185	**Error message** Selected TCPM mode cannot be combined with simultaneous turning **Cause of error** The simultaneous turning cycle does not support the programmed TCPM mode **Error correction** The following TCPM parameters must be set: - AXIS POS (coordinates = nominal position), - PATHCTRL AXIS (interpolation = nominal position) - REFPNT CENTER-CENTER or REFPNT TIP-CENTER (tool preset)
1A0-0188	**Error message** Function not permitted **Cause of error** - FUNCTION FACINGHEAD is programmed with active tool compensation regarding the workpiece coordinate system. - FUNCTION FACINGHEAD is not allowed in combination with FUNCTION TURNDATA CORR-WCS. **Error correction** Deactivate tool compensation with regard to the workpiece coordinate system.

Error number	Description
1A0-0189	**Error message** Impermissible interpolation of rotary axes **Cause of error** - A rotary axis was programmed that was deselected with M138 or in the machine parameter CfgAxisPropKin/MP_rotAxisForKinCalc = FALSE but, according to the machine parameter CfgAxisPropKin/MP_paraxComp = Display, must be considered. - This movement cannot be interpreted with TCPM movements. **Error correction** - Check the NC program and edit it if necessary - Inform your service agency
1A0-018A	**Error message** No physical axis available for handwheel superimpositioning **Cause of error** Handwheel superimpositioning in an axis that is not in the current kinematics **Error correction** - Deactivate the handwheel superimpositioning - Check the machine configuration - Inform your service agency
1A0-018B	**Error message** Contour preparation not possible **Cause of error** An internal error occurred while preparing the contour for the loaded noncircular program, which can therefore not be executed. **Error correction** Inform your service agency
1A0-018C	**Error message** Contour preparation cannot process a noncircular program **Cause of error** The noncircular program cannot be processed by the contour preparation. Possible causes: - The program contains contour elements whose length is extremely short (length < 1pm) - The parameter "F effective as C feed rate" is set in the program and the C axis reverses direction during the program or its speed (briefly) drops to zero **Error correction** - Edit the NC program - Inform your service agency

Error number	Description
1A0-018E	**Error message** Simultaneous turning: Pre-positioning **Cause of error** The current inclination of the tool is outside the programmed angle range. **Error correction** Adjust the inclination angle of the tool accordingly before calling the cycle.
1A0-018F	**Error message** Inclination angles could not be reached **Cause of error** The desired inclination angles are outside the valid inclination range. **Error correction** Adjust the inclination angle range or the desired inclination angle at the beginning or end of the contour.
1A0-0190	**Error message** Axis of a branched kinematics path used **Cause of error** An axis was used that is in a kinematics path that is currently branched. The following uses are not possible for such an axis: - Positioning with the PLANE function - Selecting the axis with M138 - Positioning within an LN block - Positioning within a CP block **Error correction** Edit the NC program
1A0-0191	**Error message** Impermissible axis was programmed **Cause of error** You have programmed an axis that is configured as a spindle in the selected kinematics model. **Error correction** Edit the NC program
1A0-0192	**Error message** Faulty turning tool data **Cause of error** The lathe tool is incorrectly defined. It does not have a permissible type. **Error correction** Correct the type of the turning tool

Error number	Description
1A0-0194	**Error message** The limit switches of a modulo axis are invalid **Cause of error** Please note that the following conditions apply for the limit switches/protection zones of modulo axes: - The lower limit must be greater than -360° and less than +360° - The upper limit must not be negative and must be less than +360° - The lower limit must not be greater than the upper limit - The lower limit and upper limit must be less than 360° apart Please also ensure that the set limit switches and protection zone result in a clearly defined traverse range. A missing or doubled overlap of the two ranges is not permitted. **Error correction** Correct an incorrectly set protection zone or incorrectly configured limit switches.
1A0-0195	**Error message** Faulty tool data for simultaneous turning cycle **Cause of error** The tool contour consisting of cutter and holder could not be determined. **Error correction** - TO, ZL, XL and ORI must agree with the tool holder geometry in KINEMATIC. - The faulty contours were stored in TNC:\system\Toolkinematics\
1A0-0196	**Error message** TCPM: PATHCTRL VECTOR not possible **Cause of error** You programmed TCPM with PATHCTRL VECTOR so that the tool orientation lies in the same plane during the complete movement from the start point to the end point. The current rotary axes in conjunction with the programmed start orientation and end orientation do not permit smooth motion. **Error correction** - Prefer PATHCHTRL AXIS. PATHCHTRL VECTOR is only useful during peripheral milling or if large changes of angle are programmed. - Program an additional NC data point in the symmetry position (pole) - Edit the NC program

Error number	Description
1A0-0198	**Error message** TCPM: PATHCTRL VECTOR not possible **Cause of error** You programmed TCPM with PATHCTRL VECTOR so that the tool orientation lies in the same plane during the complete movement from the start point to the end point. This is not possible because positions that cannot be approached are located along the path between the start orientation and the end orientation (e.g., limit switches or kinematic limitations). **Error correction** - If the fault is not due to a limit switch, prefer PATHCHTRL AXIS. PATHCHTRL VECTOR is only useful during peripheral milling or if large changes of angle are programmed. - Edit the NC program
1A0-0199	**Error message** TCPM: PATHCTRL VECTOR not possible **Cause of error** You programmed TCPM with PATHCTRL VECTOR so that the tool orientation lies in the same plane during the complete movement from the start point to the end point. This is not possible because a rotary axis that was not selected with M138 or a linear secondary axis was programmed. **Error correction** Edit the NC program.
1A0-019A	**Error message** Coordinate transformation not allowed in dressing mode **Cause of error** You attempted to switch to dressing mode even though a coordinate transformation (datum shift, rotation, mirroring, and/or scaling) is active. **Error correction** Deactivate the coordinate transformation (datum shift, rotation, mirroring, and/or scaling) before switching to dressing mode.
1A0-019B	**Error message** Tool-carrier kinematics not allowed **Cause of error** Grinding wheels with tool-carrier kinematics cannot be dressed. **Error correction** - For the grinding wheel to be dressed, delete the entry under "KINEMATIC" in the tool table - Inform your service agency

Error number	Description
1A0-019C	**Error message** Plunging not possible at position (%1, %2) **Cause of error** A pocket cannot be machined since plunging is not possible with this tool radius. **Error correction** - Use a smaller tool - Rework with a smaller tool
1A0-019D	**Error message** Limitation cannot be defined with Cycle 14 **Cause of error** The first defined contour cannot be interpreted as a border if Cycle 14 was used to define it. **Error correction** - Define contours with CONTOUR DEF or - Use the first contour as a pocket and set parameter Q569 to 0 in Cycle 271
1A0-019F	**Error message** Pocket not supported after an "open frame" **Cause of error** In the definition of the contours, a pocket (P2) is defined after a "bounding box". A "bounding box" must be followed by an island (I2). **Error correction** - Do not define a "bounding box" in Cycle 271 if a closed pocket is to be machined. - Use CONTOUR DEF to define an island after the "bounding box" if an open pocket is to be machined. - Refer to the User's Manual for more documentation.
1A0-01A0	**Error message** Plunging depth too small **Cause of error** You programmed too low a depth in Cycle 271. **Error correction** NC-Programm anpassen
1A0-01A1	**Error message** No technology data record available for contour machining **Cause of error** A Cycle 271 must be programmed before every fixed cycle 272, 273, or 274. **Error correction** - Adapt the NC program - Program Cycle 271

Error number	Description
1A0-01A3	**Error message** Plunging depth too small **Cause of error** You defined the plunging depth Q238 to be less than 0.1 mm in Cycle 274. **Error correction** - Adapt the NC program - Define the plunging depth Q238 to be greater than 0.1 mm
1A0-01A4	**Error message** Tool radius too small **Cause of error** The tool radius of the current tool is too small. **Error correction** Select a larger tool
1A0-01A5	**Error message** Roughing tool not defined **Cause of error** A roughing tool must be defined for each fixed cycle 273 and 274. **Error correction** - Adapt the NC program - Call Cycle 272 - Reference a roughing tool in parameter Q438
1A0-01A6	**Error message** Parameter 'Feed per revolution Q436' incorrectly defined **Cause of error** You entered the value 0 for the infeed per revolution parameter Q436. **Error correction** Check and correct the value in Q436
1A0-01A7	**Error message** Impermissible NC block in contour **Cause of error** This NC block is not permitted in a contour (e.g. APPR or DEP blocks, LN blocks,...) **Error correction** Edit the contour

Error number	Description
1A0-01A8	**Error message** Erroneous description of fixture in file %1 **Cause of error** The description of the fixture in the given file is faulty or the file does not exist. **Error correction** - Check the description of the fixture in the given file and correct it if necessary - Reset the fixture with FIXTURE RESET ALL - Load a valid fixture with FIXTURE SELECT - Inform your service agency
1A0-01A9	**Error message** SW limit switch for modulo axes faulty **Cause of error** Movements of modulo axes with software limit switches are not permitted in this version of the control software. **Error correction** - Edit the NC program - Do not configure limit switches for a modulo axis - Do not configure the axis as a modulo axis - Install a newer version of the NC software
1A0-01AA	**Error message** Traverse limits for modulo axes faulty **Cause of error** Movements of modulo axes with traverse limits are not permitted in this version of the control software. **Error correction** - Edit the NC program - Do not configure traverse limits for a modulo axis - Do not configure the axis as a modulo axis - Install a newer version of the NC software
1A0-01AB	**Error message** Fixture not permitted **Cause of error** Fixtures are not permitted in dressing mode. **Error correction** - Remove the entry CfgKinFixSocket from the active kinematics configuration - Inform your service agency

Error number	Description
1A0-01AC	**Error message** Configuration not suited for cylinder surface machining **Cause of error** The first machine axis under the table must be a modulo rotary axis. **Error correction** - Check the axis configuration - Inform your machine tool builder
1A0-01AD	**Error message** Fixtures ignored in "%1" **Cause of error** The indicated kinematic model has no insertion point for fixtures. **Error correction** - Adjust the machine configuration to include fixtures. To do so, please contact your machine tool builder. Please note: If you use the indicated kinematic model, the machining operation will be executed without the desired fixture.
1A0-01AE	**Error message** Island not permitted directly after a bounding block **Cause of error** In the definition of the contours, an island (I2) is defined after a "bounding block". A "bounding block" must be followed by a pocket (P2). **Error correction** - Do not define a "bounding block" in Cycle 271 if a closed pocket or stud is to be machined - Use CONTOUR DEF to define a pocket after the "bounding block" if an open pocket is to be machined. - Refer to the User's Manual for more documentation
1A0-01AF	**Error message** Invalid tool technology data **Cause of error** The tool radius is the sum of the values R and DR from the tool table; in some cases a programmed oversize has been added. If the width of the indexable insert (RCUTS) equals this tool radius, then the plunge angle (ANGLE) must be 90. **Error correction** Check the tool data and correct them if required.

Error number	Description
1A0-01B1	**Error message** Width of the indexable insert is too large **Cause of error** The tool radius results from the sum of the values R and DR from the tool table; in some cases a programmed oversize has been added. The width of the indexable insert (RCUTS) must not exceed 95% of the tool radius. **Error correction** Check the tool data and correct them if required.
1A0-01B2	**Error message** Usable length of the tool used is too small **Cause of error** The useful length of the tool being used (column LU in the tool table) is less than the machining depth programmed in the cycle. **Error correction** - Use a tool with a greater useful length
1A0-01B3	**Error message** Polar kinematics cannot be activated **Cause of error** The polar kinematics could not be activated with the programmed axes and selected solution. **Error correction** Check the selected axes and the solution: - The axes must span the three-dimensional space - The rotary axis must be built onto the table side and configured as a modulo axis (CfgAxis/isModulo = TRUE) - Exactly one rotary axis must be selected - It must be possible to reach the selected solution from the current position (MODE_POS: machine is at a positive value of the radial axis, MODE_NEG: machine is at a negative value of the radial axis)
1A0-01B4	**Error message** Polar kinematics: TCPM not allowed **Cause of error** TCPM (M128) is not allowed with active polar kinematics. **Error correction** Check the NC program and adapt it if necessary

Error number	Description
1A0-01B5	**Error message** Polar kinematics: transformation not allowed **Cause of error** Certain transformations are not permitted with active polar kinematics: - Tilt the working plane **Error correction** Check the NC program and adapt it if necessary
1A0-01B6	**Error message** Polar kinematics not possible when stretch filter is active **Cause of error** Polar kinematics cannot be activated if a "Stretch Filter" is active. **Error correction** - Check the entry under CfgStrechFilter and change it if required - Inform your service agency
1A0-01B7	**Error message** Five-axis machining is not allowed with active polar kinematics **Cause of error** Programming of linear- and rotary-axis movements in one NC block is not not permitted with active polar kinematics. **Error correction** Edit the NC program.
1A0-01B8	**Error message** Handwheel superimp. not allowed with active polar kinematics **Cause of error** Handwheel superimpositioning is not permitted with active polar kinematics **Error correction** - Deactivate handwheel superimpositioning - Deactivate polar kinematics
1A0-01B9	**Error message** Polar kinematics: combination with M91 not allowed **Cause of error** The combination of active polar kinematics and M91 is not permitted. **Error correction** Check the NC program and adapt it if necessary

Error number	Description
1A0-01BA	**Error message** Alternating machining not permitted for a closed contour **Cause of error** A value of 0 for Q15 (alternating machining direction) is not supported for a closed contour. **Error correction** Change the value for Q15 to +1 (climb) or -1 (up-cut).
1A0-F302	**Error message** APPRLT not permitted for a closed contour **Cause of error** Approaching with APPRLT is not supported for a closed contour. **Error correction** In Cycle 270 set the input parameter Q390 to 1 (APPRCT) or 3 (APPRLN).
1A0-F303	**Error message** No contours to be machined **Cause of error** After internal resolution of the contours, no (sub)contours that can be machined with OCM remain. Please note: - Pockets that are narrower than 2*R*(1+Q578) cannot be machined, due to the rounding arcs of inside corners. - Depending on R and RCUTS, no plunging is possible in narrow pockets. **Error correction** Ensure that the programmed contours are sufficiently wide, particularly concerning the dimensions stated above.
1A0-F304	**Error message** The depth will not be finished without an allowance **Cause of error** The depth will not be finished as long as no allowance is programmed for the depth (Q369). **Error correction** When defining the contour data, program an allowance for the depth in Q369.
1A1-000C	**Error message** Selected kinematics not defined **Cause of error** - Attempt to use nonexisting kinematics **Error correction** - Expand the kinematics configuration. - Change the cycle. - Inform your service agency.

Error number	Description
1A1-000D	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-000E	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-000F	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-0010	**Error message** Limit switch %1%2 CC limit switch %2 + **Cause of error** The calculated tool path exceeds the machine's positive traverse limits. The current machine setting was presumably not used and the workpiece is therefore in the wrong position in the working space. The positive software limit switch is defined with the configuration datum CfgPositionLimits->swLimitSwitchPos. **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point.

Error number	Description
1A1-0011	**Error message** Limit switch %1%2 CC limit switch %2 - **Cause of error** The calculated tool path exceeds the machine's negative traverse limits. The current machine setting was presumably not used and the workpiece is therefore in the wrong position in the working space. The negative software limit switch is defined with the configuration datum CfgPositionLimits->swLimitSwitchNeg. **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point.
1A1-0012	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-0013	**Error message** Kinematics configuration faulty %1 **Cause of error** Fewer than 3 translation axes are configured in the machine kinematics. **Error correction** - Edit the configuration of the machine kinematics - Check the number of axes in the kinematic model that is defined in the config object CfgProgAxis as MainLinCoord type - When using the FUNCTION PARAXMODE: Check the number and type of axes that you have programmed in this function - Inform your service agency

Error number	Description
1A1-0014	**Error message** Kinematics configuration faulty %1 **Cause of error** More than 3 translation axes are configured in the machine kinematics. **Error correction** - Edit the configuration of the machine kinematics - Check the number of axes in the kinematic model that is defined in the config object CfgProgAxis as MainLinCoord type - When using the FUNCTION PARAXMODE: Check the number and type of axes that you have programmed in this function - Inform your service agency
1A1-0015	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-0016	**Error message** Program position not accessible **Cause of error** The machine cannot reach all points in the space. The three linear axes with which the control moves to programmed positions all lie in one plane. Possible causes: - With FUNCTION PARAXMODE you selected three axes that lie in one plane - A linear axis is mounted on a rotary axes; the rotary axis has tilted the linear axis into the plane of the two other linear axes **Error correction** Edit the NC program
1A1-0017	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency. - Edit the cycles.

Error number	Description
1A1-0018	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-0019	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-001A	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-001B	**Error message** No accuracy specified for calculating the kinematic compen-sating movement. **Cause of error** - Accuracy data missing for calculation of the kinematic compensation movement **Error correction** - Edit the cycle.
1A1-001C	**Error message** Rotary axis undefined **Cause of error** - Machine kinematics incorrectly configured - Wrong kinematics selected **Error correction** - Edit the configuration of the machine kinematics. - Edit the cycle. - Inform you service agency.

Error number	Description
1A1-001D	**Error message** Kinematics configuration faulty %1 **Cause of error** - Machine kinematics incorrectly configured **Error correction** - Edit the configuration of the machine kinematics. - Inform your service agency.
1A1-001E	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-0022	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Choose a kinematic configuration with orthagonally arranged axes - Inform your service agency
1A1-0023	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-0024	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.

Error number	Description
1A1-0025	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-0026	**Error message** Function not yet implemented: %1 **Cause of error** - You tried to use a non-implemented function. **Error correction** - Edit the NC program
1A1-0027	**Error message** Type of grinding wheel compensation not defined **Cause of error** You have not specified the grinding wheel edge to be compensated. **Error correction** - Edit the cycle
1A1-0028	**Error message** Axis cannot be moved! %1 **Cause of error** You tried to move an axis that cannot be moved by the NC, such as a display axis, which is only displayed. **Error correction** - Check the NC program - Select suitable machine kinematics (polar)
1A1-0029	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.

Error number	Description
1A1-002A	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-002B	**Error message** Kinematics configuration faulty %1 **Cause of error** - System error - Incorrect kinematics **Error correction** - Edit the configuration of the machine kinematics. - Inform your service agency.
1A1-003B	**Error message** Grinding wheel geometry incorrect Negative value %1 in the grinding wheel parameters **Cause of error** - Incorrect parameter settings of grinding wheel geometry **Error correction** - Correct the parameters of the grinding wheel geometry.
1A1-003C	**Error message** Grinding wheel geometry incorrect Negative value for %1 in the grinding wheel parameters **Cause of error** - Incorrect parameter settings of grinding wheel geometry **Error correction** - Correct the parameters of the grinding wheel geometry.
1A1-003D	**Error message** Grinding wheel geometry incorrect Angle %1 in the grinding wheel parameters too small **Cause of error** - Incorrect parameter settings of grinding wheel geometry **Error correction** - Correct the parameters of the grinding wheel geometry.

Error number	Description
1A1-003E	**Error message** Grinding wheel geometry incorrect Negative edge length in the grinding wheel geometry **Cause of error** - Incorrect parameter settings of grinding wheel geometry **Error correction** - Correct the parameters of the grinding wheel geometry.
1A1-003F	**Error message** Grinding wheel geometry incorrect Missing parameter %1 in grinding wheel parameters **Cause of error** - Incorrect parameter settings of grinding wheel geometry **Error correction** - Correct the parameters of the grinding wheel geometry.
1A1-0040	**Error message** System error in the geometry chain: %1 **Cause of error** - System error **Error correction** - Inform your service agency.
1A1-0042	**Error message** Kinematics configuration faulty %1 **Cause of error** Invalid attribute or list element in a view message, caused by incorrect key entries in configuration messages. Because of this, the configuration server overlooks entities, which is why elements of the output lists in the view message are set to invalid. **Error correction** - Correct the kinematic configuration. - Inform your service agency.
1A1-0043	**Error message** Kinematics configuration faulty %1 **Cause of error** Cause of error is explicitly described in the error text. **Error correction** Inform your service agency.

Error number	Description
1A1-0044	**Error message** Setting of software limits for roll-over axis not permitted **Cause of error** You tried to set values for software limit switches on a roll-over axis. **Error correction** - Change the configuration - Edit the cycle
1A1-0045	**Error message** Vertical axis direction not possible **Cause of error** It is not possible to orient the tool axis orthogonally to the working plane that you defined. **Error correction** - Edit the NC program - If possible, clamp the tool differently - Where applicable, change the limit switch setting of the rotary axes
1A1-0046	**Error message** Control cannot calculate the circle tangent **Cause of error** You defined a cirle with the radius 0. **Error correction** Edit the NC program
1A1-0047	**Error message** The control cannot change the tool orientation because no rotary axes are defined **Cause of error** Rotary axes are not defined to allow changing the tool orientation. **Error correction** - Edit the NC program - Rebuild the machine - Configure the kinematics with rotary axes
1A1-0048	**Error message** Error in the kinematic configuration: %1 **Cause of error** Indicated in English in the additional text **Error correction** - Edit the kinematic configuration - Inform your service agency

Error number	Description
1A1-0049	**Error message** No axis found for tool length compensation. **Cause of error** Possible causes: there is no axis or no clearly definable axis that can compensate the tool length. **Error correction** - Change the contour configuration - Inform your service agency
1A1-004A	**Error message** Too many axes to be interpolated **Cause of error** The maximum allowed number of simultaneously moving axes was exceeded. (In the export version the maximum is 4 axes.) **Error correction** Check the NC program
1A1-004B	**Error message** Kinematics configuration faulty **Cause of error** More than 3 translation axes are configured in the machine kinematics. **Error correction** - Edit the configuration of the machine kinematics - Check the number of axes in the kinematic model that are defined in the config object CfgAxis in the Parameter specCoordSys as additional linear axes. Together with the axes in the kinematic model, which are defined in the config object CfgProgAxis as the MainLinCoord type, exactly 3 axes must be available for the machine kinematics. - When using the FUNCTION PARAXMODE: Check the number and type of axes that you have programmed in this function - Inform your service agency
1A1-004C	**Error message** Thread cutting: Direction reversal not allowed! **Cause of error** A direction reversal of the thread reference axis is not allowed. **Error correction** - You must not change the direction of the thread reference axis. - Edit the NC program.

Error number	Description
1A1-004D	**Error message** Non-interpolating axis is exceeding the traverse range! **Cause of error** A non-interpolating axis is exceeding the traverse range limits. **Error correction** Reduce the programmed path of the non-interpolating axis.
1A1-004E	**Error message** Distance is too short to accelerate non-interpolating axis! **Cause of error** A non-interpolating axis is exceeding the maximum acceleration! **Error correction** Extend the programmed path of the non-interpolating axis.
1A1-004F	**Error message** This area cannot be machined! **Cause of error** No traverse permitted with polar kinematics in the programmed range. **Error correction** Check the height difference of the point or the position of the fixed axis. Check whether the machining operation is permitted at the pole.
1A1-0050	**Error message** Limit switch with handwheel superimpositioning %1%2 **Cause of error** The calculated tool path exceeds the machine's positive traverse limits. The current machine setting was presumably not adopted and the workpiece is therefore in the wrong position in the working space. M118 limit switch **Error correction** Reduce the handwheel traverse range (M118)
1A1-0051	**Error message** Wrong kinematic model for FACING HEAD POS **Cause of error** The active kinematic model had not facing slide. FACING HEAD POS is allowed only with facing-slide kinematics. **Error correction** Insert the facing slide and switch the kinematics

Error number	Description
1A1-0052	**Error message** Tool inclination cannot be calculated **Cause of error** There are too many or too few rotary axes present in order to calculate the tool angle of inclination **Error correction** - Use M138 to select or deselect the rotary axes - Check the configuration of the tool spindle, particularly CfgAxisPropKin/rotAxisForKinCalc - Contact your machine tool builder
1A1-0053	**Error message** Tool inclination cannot be calculated **Cause of error** The orientation of the indexable insert of the turning tool is not permissible. **Error correction** The plane of the indexable insert must be parallel or perpendicular to the tool spindle: - Check the tool data - Check the kinematics configuration, particularly the transformations (CfgKinSimpleTrans) between the tool spindle and tool - Contact your machine tool builder
1A1-0054	**Error message** Tool inclination cannot be calculated **Cause of error** Incorrect orientation of the selected rotary axes. Possible causes: - The turning spindle is parallel to the selected tilting axis - The tool direction is parallel to the selected tilting axis - The programmed inclination is not possible with the present device **Error correction** - Check the programmed inclination - Use M138 to select a different tilting axis - Check the kinematics configuration - Check the configuration of the tool spindle, particularly CfgAxisPropKin/rotAxisForKinCalc - Contact your machine tool builder
1A2-000A	**Error message** System error in the calculation of transformation: %1 **Cause of error** Ist im Fehlertext direkt angegeben **Error correction** - Kundendienst benachrichtigen

Error number	Description
1A2-000B	**Error message** Axis programmed more than once in PRESET command **Cause of error** Sie haben im PRESET-Befehl mehrfach dieselbe Achse programmiert. **Error correction** - NC-Programm bzw. Zyklus ändern
1C7-01F6	**Error message** Data record for the FS configuration of the SPLC program **Cause of error** Data record for the configuration of functional safety for the SPLC program **Error correction**
1C7-0205	**Error message** Max. time until the test of the motor holding-brakes **Cause of error** Axis-specific time monitoring for the execution of the brake test. - 0: No time monitoring through the SKERN **Error correction**
1C7-0206	**Error message** Maximum path with SS2 reaction **Cause of error** Maximum permissible path or spindle revolutions for SS2 reaction in SOM2 operating mode **Error correction**
1C7-0207	**Error message** Maximum path with SS2 reaction **Cause of error** Maximum permissible path or revolutions of the spindle with SS2 reaction in SOM3 operating mode **Error correction**
1C7-0208	**Error message** Maximum path with SS2 reaction **Cause of error** Maximum permissible path or spindle revolutions for SS2 reaction in SOM4 operating mode **Error correction**

Error number	Description
1C7-021B	**Error message** MC drives cannot be switched on: NN_GenSafe = 0 **Cause of error** - SPLC interface signal NN_GenSafe = 0. It is therefore impossible to switch on the drives. - SPLC program does not set the interface signal. - The machine parameter MP_skipEmStopTest has been set. **Error correction** - Check the SPLC program. - Check the entry in MP_skipEmStopTest. - Inform your service agency.
1C7-0255	**Error message** Following error monitoring for RTC coupled axes **Cause of error** Position monitoring for coupled axes. If the axis cannot follow the RTC specification and the position difference exceeds this value, an EMERGENCY STOP reaction is triggered. You can find information on braking the drives during an EMERGENCY STOP in the Technical Manual for your control. The settings in posTolerance apply only during active RTC and are independent of the settings in CfgPosControl. **Error correction**
1C7-025F	**Error message** Default HSCI data rate **Cause of error** Enter the desired HSCI data rate. With the “automatic” option, the control automatically selects the highest possible data rate. If there are devices or cables in the HSCI system that are suited for only limited data rates, a manual preselection might be necessary. **Error correction**
1C7-0268	**Error message** Permit handwh. superimposition of rotary axes only with TCPM **Cause of error** **Error correction**

Error number	Description
1C7-0350	**Error message** Mode of nominal/actual value monitoring (optional) **Cause of error** The parameter specifies the type of nominal-actual-value monitoring: - speedAndPosCompDefault: With FS spindles, the nominal-actual-value comparison is always performed for rotational speeds; with position-controlled feed axes when the guard doors are open for positions, and when the guard doors are closed for speeds. - speedAndPosCompReduced: With FS spindles, the nominal-actual-value comparison is always performed for rotational speeds; with position-controlled axes when the guard doors are open for positions and otherwise for speeds - noComp: The nominal-actual-value comparison is inactive, so neither speeds nor positions are checked - speedComp: With FS spindles, the nominal-actual-value comparison is performed for rotational speeds, and with feed axes for positions. **Error correction**
1C9-006B	**Error message** Description of a tool carrier **Cause of error** Describe here a tool carrier. **Error correction**
200-0001	**Error message** Calculator **Cause of error** **Error correction**
200-0017	**Error message** Configuration of '%1' incorrect **Cause of error** The configuration data are incomplete or contain invalid values. **Error correction** Check the configuration data.

Error number	Description
200-0018	**Error message** Configuration for axis %1 invalid **Cause of error** The configuration data for the specified axis are incomplete or contain invalid values. **Error correction** Check the configuration datum "CfgProgAxis" for this axis.
200-0019	**Error message** Configured name for axis %1 invalid **Cause of error** The configured name for an axis with the configured properties is not permissible or is already assigned to another axis. **Error correction** Change the configuration datum "CfgProgAxis/axName"
200-001A	**Error message** Configured index for axis %1 invalid **Cause of error** The configured index for an axis with the configured properties is not permissible or is already assigned to another axis. **Error correction** Change the configuration datum "CfgProgAxis/index"
200-001B	**Error message** Configured direction for axis %1 invalid **Cause of error** None of the directions XAxis, YAxis or ZAxis is configured for the axis. **Error correction** Change the configuration datum "CfgProgAxis/dir"
200-001C	**Error message** Cycle %1 already defined **Cause of error** In CycleDesign, the same number or G number was assigned more than once for a cycle or query cycle. **Error correction** In CycleDesign, change the number of one of the cycles or its G number.

Error number	Description
200-001D	**Error message** Unable to open configuration server queue **Cause of error** Error in system-inherent communication **Error correction** Inform your service agency
200-001E	**Error message** Unable to read configuration data '%1' **Cause of error** Error in system-inherent communication **Error correction** Inform your service agency.
200-001F	**Error message** No programmable axes defined for the editor **Cause of error** The configuration is faulty: CfgEditorSettings/useProgAxes is set so that the programmable axes defined through CfgChannelAxes/progAxis are to be used for the editor. CfgChannelAxes/progAxis is empty, however. **Error correction** Correct the configuration: CfgEditorSettings/useProgAxes
200-0020	**Error message** NC program is incomplete **Cause of error** No valid program end found: - File was not transmitted to the control completely - File corrupted during editing with a text editor - Error in the file system **Error correction** - Retransmit the file or restore it from an archive - Manually correct the file in the NC editor Note: The NC editor automatically appends a program end for visual purposes. Use 'Save as' to write this program end into the file.
201-0800	**Error message** Key non-functional Key non-functional Key non-functional Key non-functional Key non-functional Key non-functional **Cause of error** In this context the key has no function.

Error number	Description
201-0801	**Error message** Program memory exceeded **Cause of error** The NC program memory no longer suffices for part programs. **Error correction** Delete the programs that you no longer need.
201-0802	**Error message** Search address missing **Cause of error** In the NC program the original search address no longer exists. **Error correction** Abort search.
201-0803	**Error message** Entry value incorrect Entry value incorrect Entry value incorrect **Cause of error** - The value you entered is out-of-range. - Cycle 209 (ISO: 209): You entered the value 0 as infeed depth for chip breaking (Q257). **Error correction** - Enter the correct value. - Enter a value other than 0 in Q257.
201-0804	**Error message** Program not found **Cause of error** You attempted to call a program that is not stored in TNC memory. **Error correction** Edit the part program.
201-0805	**Error message** Protected File! Protected File! **Cause of error** You cannot edit or erase this program until the protection has been removed. **Error correction** Cancel the program protection.

Error number	Description
201-0806	**Error message** Block format incorrect **Cause of error** Incorrect block format in the highlighted block. **Error correction** Edit the part program.
201-0807	**Error message** Address letter already assigned **Cause of error** You used an address letter incorrectly in an ISO block. **Error correction** Edit the highlighted block.
201-0808	**Error message** Block too long DATUM SHIFT **Cause of error** **Error correction**
201-092E	**Error message** Traverse direction not defined Traverse direction not defined Traverse direction not defined **Cause of error** In a probing cycle you entered 0 for the traverse direction Q267. **Error correction** For Q267, enter either +1 (for positive traverse direction) or -1 (for negative traverse direction).
201-092F	**Error message** No datum table active No datum table active **Cause of error** Probing cycle for datum setting: You want the TNC to write the measured point into a datum table, but you have not activated a datum table in a program run mode (status M). **Error correction** In the single block or full sequence program run mode, activate the datum table into which you want the measured point to be entered.

Error number	Description
201-0930	**Error message** Position error: center in axis 1 Position error: center in axis 1 Position error: center in axis 1 **Cause of error** Probing cycle for workpiece measurement: Center of 1st axis outside of position tolerance. **Error correction** Check the workpiece and the measuring log.
201-0931	**Error message** Position error: center in axis 2 Position error: center in axis 2 Position error: center in axis 2 **Cause of error** Probing cycle for workpiece measurement: Center of 2st axis outside of position tolerance. **Error correction** Check the workpiece and the measuring log.
201-0932	**Error message** Hole diameter too small Hole diameter too small Hole diameter too small **Cause of error** Probing cycle for workpiece measurement: Hole diameter too small for tolerance. **Error correction** Check the workpiece and the measuring log.
201-0933	**Error message** Hole diameter too large Hole diameter too large Hole diameter too large **Cause of error** - Probing cycle for workpiece measurement: Hole diameter tolerance exceeded. - Cycle 208: The programmed hole diameter (Q335) cannot be machined with the active tool. **Error correction** - Check the workpiece and, if necessary, the measuring log. - Cycle 208: Use a larger tool. Hole diameter must not be larger than twice the tool diameter.

Error number	Description
201-0934	**Error message** Stud diameter too small Stud diameter too small Stud diameter too small **Cause of error** Probing cycle for workpiece measurement: Stud diameter too small for tolerance. **Error correction** Check the workpiece and the measuring log.
201-0935	**Error message** Stud diameter too large Stud diameter too large Stud diameter too large **Cause of error** Probing cycle for workpiece measurement: Stud diameter too large for tolerance. **Error correction** Check the workpiece and the measuring log.
201-0936	**Error message** Pocket too small: rework axis 1 Pocket too small: rework axis 1 Pocket too small: rework axis 1 **Cause of error** Probing cycle for workpiece measurement: Pocket length in 1st axis too small for tolerance. **Error correction** Check the workpiece and the measuring log.
201-0937	**Error message** Pocket too small: rework axis 2 Pocket too small: rework axis 2 Pocket too small: rework axis 2 **Cause of error** Probing cycle for workpiece measurement: Pocket width in 2nd axis too small for tolerance. **Error correction** Check the workpiece and the measuring log.

Error number	Description
201-0938	**Error message** Pocket too large: scrap axis 1 Pocket too large: scrap axis 1 Pocket too large: scrap axis 1 **Cause of error** Probing cycle for workpiece measurement: Pocket length in 1st axis too large for tolerance. **Error correction** Check the workpiece and the measuring log.
201-0939	**Error message** Pocket too large: scrap axis 2 Pocket too large: scrap axis 2 Pocket too large: scrap axis 2 **Cause of error** Probing cycle for workpiece measurement: Pocket width in 2nd axis too large for tolerance. **Error correction** Check the workpiece and the measuring log.
201-093A	**Error message** Stud too small: scrap axis 1 Stud too small: scrap axis 1 Stud too small: scrap axis 1 **Cause of error** Probing cycle for workpiece measurement: Stud length in 1st axis too small for tolerance. **Error correction** Check the workpiece and the measuring log.
201-093B	**Error message** Stud too small: scrap axis 2 Stud too small: scrap axis 2 Stud too small: scrap axis 2 **Cause of error** Probing cycle for workpiece measurement: Stud width in 2nd too small for tolerance. **Error correction** Check the workpiece and the measuring log.

Error number	Description
201-093C	**Error message** Stud too large: rework axis 1 Stud too large: rework axis 1 Stud too large: rework axis 1 **Cause of error** Probing cycle for workpiece measurement: Stud length in 1st axis too large for tolerance. **Error correction** Check the workpiece and the measuring log.
201-093D	**Error message** Stud too large: rework axis 2 Stud too large: rework axis 2 Stud too large: rework axis 2 **Cause of error** Probing cycle for workpiece measurement: Stud width in 2nd axis too large for tolerance. **Error correction** Check the workpiece and the measuring log.
201-093E	**Error message** Meas. cycle: length exceeds max Meas. cycle: length exceeds max Meas. cycle: length exceeds max **Cause of error** Probing cycle 425 or 427: The measured length exceeds the maximum permissible value. **Error correction** Check the workpiece and the measuring log.
201-093F	**Error message** Meas. cycle: length below min Meas. cycle: length below min Meas. cycle: length below min **Cause of error** Probing cycle 425 or 427: The measured length is below the minimum permissible value. **Error correction** Check the workpiece and the measuring log.

Error number	Description
201-0940	**Error message** TCHPROBE 426: length exceeds max TCHPROBE 426: length exceeds max TCHPROBE 426: length exceeds max **Cause of error** Probing cycle 426: The measured length exceeds the maximum permissible value. **Error correction** Check the workpiece and the measuring log.
201-0941	**Error message** TCHPROBE 426: length below min TCHPROBE 426: length below min TCHPROBE 426: length below min **Cause of error** Probing cycle 426: The measured length is below the minimum permissible value. **Error correction** Check the workpiece and the measuring log.
201-0942	**Error message** TCHPROBE 430: diameter too large TCHPROBE 430: diameter too large TCHPROBE 430: diameter too large **Cause of error** Probing cycle 430: The measured bolt-hole-circle diameter exceeds the maximum permissible value. **Error correction** Check the workpiece and the measuring log.
201-0943	**Error message** TCHPROBE 430: diameter too small TCHPROBE 430: diameter too small TCHPROBE 430: diameter too small **Cause of error** Probing cycle 430: The measured bolt-hole-circle diameter is below the minimum permissible value. **Error correction** Check the workpiece and the measuring log.
201-094F	**Error message** Incomplete cycle was deleted **Cause of error** Informational message that the TNC has erased an incomplete cycle. **Error correction**

Error number	Description
201-0950	**Error message** Intermediate memory empty **Cause of error** You attempted to insert a block from an empty intermediate memory. **Error correction** before trying to insert a block from intermediate memory, put the block into memory by: - using the DEL key to delete the block to be copied, or - placing the editing cursor into the block to be copied
201-0951	**Error message** No permission to write **Cause of error** You have selected a write-protected file for editing. **Error correction** Before editing, enter the code number 86357 to cancel the write protection.
201-0952	**Error message** To delete entire context: NO ENT **Cause of error** During editing you attempted to delete a word that is a required element of a function. **Error correction** Press NO ENT to delete the entire function, or press END to cancel.
201-0953	**Error message** Impermissible change of context **Cause of error** You attempted to open another context within a context dialog sequence. **Error correction** Continue the dialog, or completely erase the block and enter a new context.
201-0954	**Error message** No polar coordinates possible No polar coordinates possible **Cause of error** You pressed the P key to enter polar coordinates. Polar coordinates are not programmable for the active function. **Error correction** Enter Cartesian coordinates to program the active function, or use a function that permits polar coordinate input.

Error number	Description
201-0955	**Error message** To change context: press ENT **Cause of error** You attempted to change a context initiator to which other elements in the current block belong. **Error correction** First delete the elements, then change the context initiator.
201-0956	**Error message** Input as context not permitted **Cause of error** You entered a function that cannot initiate a context. **Error correction** Enter only permissible functions.
201-0957	**Error message** Check parentheses for pairs **Cause of error** You attempted to end a Q-parameter block containing an odd number of parentheses. Parentheses can be programmed only in pairs. **Error correction** Enter the missing parentheses.
201-099D	**Error message** Too many decimal places **Cause of error** **Error correction**
201-099E	**Error message** File name not equal program name **Cause of error** **Error correction**
201-099F	**Error message** Context code name unknown **Cause of error** **Error correction**
201-09A0	**Error message** Numerical value out of range **Cause of error** **Error correction**

Error number	Description
201-09A1	**Error message** Syntax error **Cause of error** Syntax error **Error correction**
201-09A2	**Error message** NC language unknown **Cause of error** **Error correction**
201-09A3	**Error message** File access not possible **Cause of error** **Error correction**
201-09A5	**Error message** Required value missing **Cause of error** You attempted to save an NC block although you have not entered all values required for the elements programmed in the block. **Error correction** Enter the NC block with all required data. Refer to the User's Manual if required.
201-09A8	**Error message** Table data unknown **Cause of error** Table data unknown **Error correction**
201-09A9	**Error message** Syntax error in binary record **Cause of error** Syntax error in binary record **Error correction**
201-0A1F	**Error message** Entry character not allowed **Cause of error** You attempted to enter a character that is not allowed in the entry box. **Error correction** Edit the NC program.

Error number	Description
201-0A20	**Error message** Lowercase letter not allowed **Cause of error** You attempted to enter a lowercase letter. **Error correction** Edit the NC program. Use only uppercase letters in the entry box.
201-0A21	**Error message** Letter not allowed **Cause of error** You attempted to enter a letter. **Error correction** Edit the NC program. Use only numerals in the entry box.
201-0A22	**Error message** Numeral not allowed **Cause of error** You attempted to enter a numeral. **Error correction** Edit the NC program. Use only letters in the entry box.
201-0A51	**Error message** Reference to block %.6s:no DEL **Cause of error** FK programming: You have attempted to delete a part program block to which another block refers. **Error correction** First edit the referring block, then delete the reference
201-0A52	**Error message** FK reference to current block **Cause of error** You attempted in an FK program to delete a block to which another part of the program makes a reference. **Error correction** Change the FK reference.

Error number	Description
201-0A55	**Error message** No axes selected **Cause of error** In the MOD settings of the machining modes, no axes were selected for L-block generation. **Error correction** In the MOD settings, enter the axes whose positions are to be put into an L block when the "actual-position-capture key" is pressed.
201-0A6F	**Error message** Too many characters **Cause of error** You exceeded the maximum permissible number of charac-ters for the active input box. **Error correction** Enter fewer characters.
201-0A70	**Error message** Value out of input range **Cause of error** You attempted to enter a numerical value that lies outside the permissible range. **Error correction** Comply with the permissible input range.
201-0A71	**Error message** Too many decimal places **Cause of error** You attempted to enter a value that exceeds the permissible number of decimal places. **Error correction** Comply with the permissible input range.
201-0A72	**Error message** Algebraic sign not allowed **Cause of error** You tried to enter an algebraic sign using the -/+ key. **Error correction** - The -/+ key is non-functional. - It is not necessary/possible to enter the algebraic sign.

Error number	Description
201-0A73	**Error message** Only integers permitted **Cause of error** You tried to enter a number with decimal places. **Error correction** - The , key is non-functional. - It is not necessary/possible to enter a decimal place.
201-0A74	**Error message** Q not allowed: enter a number **Cause of error** You attempted to use a Q parameter in the active input box. **Error correction** Enter a numerical value.
201-0A75	**Error message** Incremental input not allowed Incremental input not allowed **Cause of error** You attempted to enter an incremental value by pressing the I key. **Error correction** Enter an absolute value.
201-0A76	**Error message** Too many M functions **Cause of error** Too many M functions for one NC block. **Error correction** Use no more than two M functions per NC block.
201-0A77	**Error message** Too many axes programmed **Cause of error** - An NC block has more than the allowed number of simultaneously movable axes. - You tried to use an NC program to make a reverse program in which more than 5 axes are programmed. **Error correction** - Standard version: Program no more than 5 axes per NC block. - Export version: Program no more than 4 axes per NC block. - The source program must not include more than a total of 5 different axes.

Error number	Description
201-0A78	**Error message** Axis double programmed **Cause of error** You programmed the same axis more than once within one NC block. **Error correction** Program only different axes within one block.
201-0A79	**Error message** Element double / not allowed **Cause of error** - You used the same syntax element more than once within one NC block.- The current sequence of syntax elements in an NC block do not comply with requirements. **Error correction** - Do not program syntax elements more than once within one NC block.- Put the syntax elements into the required sequence.
201-0A7A	**Error message** Incomplete data input **Cause of error** Not all required data have been entered in an NC block. **Error correction** Add the missing information.
201-0A7B	**Error message** ISO: Block number N missing **Cause of error** An NC block in an ISO program begins without a block number N. **Error correction** Insert the block number.
201-0A7C	**Error message** Required element missing Required element missing **Cause of error** Not all required data have been entered in an NC block. **Error correction** Add the missing information.

Error number	Description
201-0A7D	**Error message** Syntax incorrect **Cause of error** An NC block contains a syntax element that requires other syntax elements. **Error correction** Correct the NC block.
201-0A9F	**Error message** Keyword unknown **Cause of error** In an NC block, you attempted to enter a word that the TNC cannot interpret. **Error correction** Enter only valid words.
201-0AA0	**Error message** Syntax element not editable **Cause of error** You attempted to edit a syntax element in an NC block. **Error correction** Enter a new NC block with a different syntax element.
201-0AA1	**Error message** PGM header not editable **Cause of error** In a program, you attempted to edit one of the blocks BEGIN PGM (ISO: %... G71), or END PGM (ISO: N99999999%...). **Error correction** The program beginning and program end must not be edited. To change the program name, use the RENAME function in the file management.
201-0AA2	**Error message** No reference system change! **Cause of error** You attempted to change the coordinate data in the present block from Cartesian to polar or vice versa. **Error correction** Move the cursor to the block initation element and press the P key to change to polar or Cartesian coordinate input.

Error number	Description
201-0AA3	**Error message** Rotary axis not permitted here **Cause of error** You programmed a rotary axis as tool axis. **Error correction** Program only linear axes in the TOOL CALL block (ISO: T..).
201-0AA4	**Error message** Incorrect block syntax **Cause of error** A part program block contains a syntax error. **Error correction** Edit the part program.
201-0AA5	**Error message** Really delete NC block? DEL! **Cause of error** Warning before deleting an NC block. **Error correction** For complete deletion of the NC block, press DEL. To abort the delete sequence, press any other key.
201-0AA6	**Error message** Axis letter not permitted **Cause of error** An attempt was made to program an axis that is not allowed for the currently active function. **Error correction** Only program permitted axes.
201-0AA7	**Error message** Cannot cancel Block Skip **Cause of error** You attempted to cancel the Block Skip funktion with the Backspace key. **Error correction** Function is only permitted if the NC block begins with / .
201-0AA8	**Error message** String incomplete **Cause of error** You attempted to enter an NC block in which a syntax element was not concluded with the required apostrophe. **Error correction** Ensure that apostrophes are entered in the correct locations. Refer to the User's Manual if required.

Error number	Description
201-0AB4	**Error message** Cannot go past edge of screen **Cause of error** You moved the display position of the current block to the edge of the screen. **Error correction** Select the display position of the current block so that it lies within the screen limits.
201-0ADF	**Error message** Actual pos. capture not possible **Cause of error** You attempted to load the actual position into the program while the tilted working plane function was active. **Error correction** The actual position can be loaded only if the tilted working plane function is not active.
201-0AFE	**Error message** Context change only on initiat.! **Cause of error** You tried to make a major change to the format of an NC block. **Error correction** Changing the NC block format is possible only if you place the cursor on the block initiator.
201-0B31	**Error message** CANCELLATION in block %u (%u %%) **Cause of error** You have aborted the search process. **Error correction** If required, restart the search function and let it terminate the search process.
201-0B67	**Error message** Line %u label name already assigned **Cause of error** You tried to assign the same label name in more than one NC block containing LBL SET. **Error correction** Use different label names.

Error number	Description
201-0B88	**Error message** No editing while PGM is running **Cause of error** - You tried to edit a program that is now being run. - You tried to edit a table that is accessed from within the program that is now being run. **Error correction** - Make changes only in the stopped condition. - Stop the program (internal stop) and reselect it with the PGM MGT key. Then edit the AFC settings.
201-0C02	**Error message** File system I/O error **Cause of error** Error during access to a file system device. **Error correction** - For TNC drives, switch the control off and on to test the drives. Contact HEIDENHAIN if the problem recurs. - For network drives, check the network connection and the computer providing the directory. - To download a table, ensure correct table contents (for redundant lines, for example).
210-0001	**Error message** End of a system file, no identifier found **Cause of error** An identifier was expected in a message file, but the file end was reached. **Error correction** Inform your service agency.
210-0002	**Error message** Identifier in system file expected **Cause of error** An identifier was expected in a message file, but a nonal-phanumeric character was read. **Error correction** Inform your service agency.
210-0003	**Error message** End of a system file, no string found **Cause of error** A string was expected in a message file, but the file end was reached. **Error correction** Inform your service agency.

Error number	Description
210-0004	**Error message** String in system file expected **Cause of error** A string beginning with " was expected in a message file, but another character was read. **Error correction** Inform your service agency.
210-0005	**Error message** Incompatible data types in system file **Cause of error** In a message file data were read that do not fit the data object to be read. **Error correction** Inform your service agency.
210-0006	**Error message** Unknown entity name in system file Message library incompatible or no entity instance implemented **Cause of error** An undefined message was read in a message file. **Error correction** Inform your service agency.
210-0007	**Error message** Identifier already assigned in system file **Cause of error** **Error correction**
210-0008	**Error message** Integral value expected in system file **Cause of error** An integer was expected in a message file. **Error correction** Inform your service agency.
210-0009	**Error message** Floating point number expected in system file **Cause of error** A floating-point number was expected in a message file. **Error correction** Inform your service agency.

Error number	Description
210-000A	**Error message** Invalid logical value in system file **Cause of error** A message file should contain a logical value (TRUE or FALSE, or a Q parameter with numerical value 0 or 1). **Error correction** Inform your service agency
210-000B	**Error message** Invalid list number in system file **Cause of error** A whole number was expected in a message file, but an undefined string or a Q parameter with invalid numerical value was read. **Error correction** Inform your service agency.
210-000C	**Error message** "(" expected in system file **Cause of error** A opening parenthesis "(" was expected in a message file. **Error correction** Inform your service agency.
210-000D	**Error message** Unexpected end of system file **Cause of error** More characters were expected in a message file, but the file end was reached. **Error correction** Inform your service agency.
210-000E	**Error message** Unknown attribute name in system file **Cause of error** In a message file an unknown message attribute was read. **Error correction** Inform your service agency.
210-000F	**Error message** Attribute already assigned in system file **Cause of error** In a message file a message attribute was read more than once. **Error correction** Inform your service agency.

Error number	Description
210-0010	**Error message** ":=" expected in system file **Cause of error** A colon and equal sign ":=" were expected in a message file. **Error correction** Inform your service agency.
210-0011	**Error message** ")" or "," expected in system file **Cause of error** A closing parenthesis ")" or a comma "," was expected in a message file. **Error correction** Inform your service agency.
210-0012	**Error message** "[" expected in system file **Cause of error** An opening square bracket "[" was expected in a message file. **Error correction** Inform your service agency.
210-0013	**Error message** End of a system file while reading a list **Cause of error** **Error correction**
210-0014	**Error message** End of a system file while reading an array **Cause of error** In a message file the file end was reached while an array was being read. **Error correction** Inform your service agency.
210-0015	**Error message** "]" or "," expected in system file **Cause of error** A closing square bracket "]" or comma "," was expected in a message file. **Error correction** Inform your service agency.

Error number	Description
210-0016	**Error message** List in system file too long **Cause of error** A message contains a list with more elements than allowed. **Error correction** Inform your service agency.
210-0017	**Error message** List in system file too short **Cause of error** A message contains a list with fewer elements than allowed. **Error correction** Inform your service agency.
210-0018	**Error message** Incorrect binary data in system file (string) **Cause of error** Error during binary transmission of a string in a message **Error correction** Inform your service agency.
210-0019	**Error message** Incorrect binary data in system file **Cause of error** An error occurred during the binary transfer of a binary number in a message. **Error correction** Inform your service agency
210-001A	**Error message** Incorrect binary data in system file (list) **Cause of error** Error during binary transmission of a list in a message. **Error correction** Inform your service agency.
210-001B	**Error message** Incorrect binary data in system file (array) **Cause of error** Error during binary transmission of an array in a message. **Error correction** Inform your service agency.

Error number	Description
210-001C	**Error message** Incorrect binary data in system file (entity) **Cause of error** Error during binary transmission of a message. **Error correction** Inform your service agency.
210-001D	**Error message** Error in system file **Cause of error** An error occurred during access to an internal list element. **Error correction** Inform your service agency
210-001E	**Error message** Invalid array index in system file **Cause of error** Access with an illegal index to an array. **Error correction** Inform your service agency.
210-001F	**Error message** Invalid Q parameter index in system file **Cause of error** An excessively large Q-parameter index is being used in a message file. **Error correction** Inform your service agency.
210-0020	**Error message** Invalid binary data in system file **Cause of error** In a message, a binary number was supposed to be read (% followed by a combination of 0 and 1). **Error correction** Inform your service agency
210-0021	**Error message** Invalid attribute name in system file **Cause of error** An undefined attribute name was sought in a message. **Error correction** Inform your service agency.

Error number	Description
210-0022	**Error message** No base type defined in system file **Cause of error** Nonavailable information about base types was requested in a message. **Error correction** Inform your service agency.
210-0023	**Error message** Error during access to system file **Cause of error** Basic reading error while reading a message file. **Error correction** Inform your service agency.
210-0024	**Error message** Insufficient memory **Cause of error** The message memory manager has no more memory. **Error correction** Inform your service agency.
210-0025	**Error message** System error: Insufficient memory **Cause of error** The message memory management is not receiving required resources from the system. **Error correction** Inform your service agency.
210-0026	**Error message** System error: File mapping **Cause of error** The message memory management could not create global buffers. **Error correction** Inform your service agency.
210-0027	**Error message** Requested memory block too large **Cause of error** An excessively large globel message buffer was requested. **Error correction** Inform your service agency.

Error number	Description
210-0028	**Error message** Invalid memory block returned **Cause of error** An invalid buffer was returned to the message memory management. **Error correction** Inform your service agency.
210-0029	**Error message** Memory block already returned **Cause of error** A buffer was returned repeatedly to the message memory management. **Error correction** Inform your service agency.
210-002A	**Error message** Missing type information in system file **Cause of error** The given message type is unknown. **Error correction** Inform your service agency.
210-002B	**Error message** Invalid attribute index in system file **Cause of error** Information was requested on a nonexistent message attribute. **Error correction** Inform your service agency.
210-002C	**Error message** Invalid supertype index in system file **Cause of error** Information was requested on a nonexistent message super-type. **Error correction** Inform your service agency.
210-002D	**Error message** Invalid function call in system file **Cause of error** A function was called that is not allowed for Q messages. **Error correction** Inform your service agency.

Error number	Description
210-002E	**Error message** Invalid Q message data in system file **Cause of error** Error in a message file while reading a Q message. **Error correction** Inform your service agency.
210-002F	**Error message** Invalid Q string **Cause of error** An excessively long string was assigned to a Q-String. **Error correction** Inform your service agency.
220-0002	**Error message** Internal software error **Cause of error** System error. The given message contains an attribute with illegal value. **Error correction** Inform your service agency.
220-0003	**Error message** Internal software error **Cause of error** Attempt to read the same measuring position more than once with the same identifier. **Error correction** Inform your service agency.
220-0004	**Error message** Internal software error Internal software error Internal software error Internal software error **Cause of error** System error **Error correction** Inform your service agency.

Error number	Description
220-000A	**Error message** Start was not executed **Cause of error** Start of an application that cannot be run together with another. There are unacknowledged errors in the error window. **Error correction** First end the application. Delete the error message.
220-000C	**Error message** Message will not be handled in its present state **Cause of error** Message will not be handled in its present state. **Error correction** None
220-000E	**Error message** Error in module configuration **Cause of error** The object requested by the configuration server could not be found. **Error correction** Edit the configuration data. Inform your service agency.
220-000F	**Error message** Error in TOOL DEF or TOOL CALL cycle **Cause of error** The TOOL DEF message was followed by an incorrect TOOL CALL message. **Error correction** Inform your service agency.
220-0010	**Error message** Internal software error **Cause of error** Internal software error **Error correction** Inform your service agency.

Error number	Description
220-0011	**Error message** Error occured in startup cycle **Cause of error** Startup cycle interrupted with error. **Error correction** Remove the cause of error and delete the error message. The cycle is restarted.
220-0013	**Error message** Configuration error occured **Cause of error** Channel name must be unambiguous **Error correction** Edit the configuration data.
220-0014	**Error message** Invalid message %1 **Cause of error** The entered message contains an attribute with invalid value. **Error correction** No further action required
220-0015	**Error message** File access not possible **Cause of error** 1.) Access to the file was denied. 2.) Another application is already writing to the file. 3.) Error in path name. 4.) Data medium full. **Error correction** 1.) Check the access rights to the file and remove any existing write protection. 2.) Close the file in the application that has locked access to the file. 3.) Correct the entered path name. 4.) Delete any unneeded files on the data medium.
220-0016	**Error message** Disk full **Cause of error** Data medium full. **Error correction** Delete any unneeded files on the data medium.

Error number	Description
220-0017	**Error message** File close failed **Cause of error** Error while closing the file. **Error correction** Ensure that the file is not being used by another application.
220-0018	**Error message** General internal communication error **Cause of error** An error has occurred in the system-inherent communication. **Error correction** Inform your service agency.
220-0019	**Error message** ClientQueue (%1) could not be opened **Cause of error** Error in system-inherent communication. No access possible to the specified queue. **Error correction** Inform your service agency.
220-001A	**Error message** Cannot write to queue '%1' **Cause of error** Error in the system-inherent communication. An error occurred while writing data to the specified queue. **Error correction** Inform your service agency.
220-001B	**Error message** Cannot close queue '%1' **Cause of error** Error in the system-inherent communication. The specified queue cannot be closed. **Error correction** Inform your service agency.

Error number	Description
220-001C	**Error message** Unknown error Unknown error **Cause of error** An unknown error occurred during the execution of a program. **Error correction** - Inform your service agency.
220-001D	**Error message** Value too small in %1-%2 **Cause of error** - The entered value is below the minimum limit value. **Error correction** - Change the value. - Check the minimum limit value.
220-001E	**Error message** Value too large in %1-%2 **Cause of error** - The entered value is above the maximum limit value. **Error correction** - Change the value. - Check the maximum limit value.
220-001F	**Error message** Value in %1 out of range **Cause of error** - The entered value is outside of the permitted value range. **Error correction** - Change the value. - Check the limit values.
220-0020	**Error message** System error in the channel object **Cause of error** System error in the channel object **Error correction** - Inform your service agency.

Error number	Description
220-0021	**Error message** Invalid FN14 function **Cause of error** - Stop error not allowed within an internal cycle - Stop error not alllowed after start via soft key **Error correction** Edit the cycle or inform your service agency or machine tool builder.
220-0022	**Error message** System error in program run: Control might be inconsistent **Cause of error** An error occurred in an internal cycle. The internal data of the control might therefore be inconsistent. **Error correction** - Shut down the control as soon as possible and restart it. Until then procede with increased caution. - Inform your service agency.
220-0023	**Error message** Format file defective **Cause of error** The outputs with FN16: F-PRINT (ISO: D16) have reached the maximum size. **Error correction** Change the format file. If necessary, output each text individually, concluding with M_CLOSE.
220-0024	**Error message** Function not available **Cause of error** During a block scan on a control without a history, an attempt was made to implement a PLC strobe with a macro. The function cannot be run on this control. **Error correction** - Change the machine configuration - Inform your service agency
220-0025	**Error message** Contradictory data during PLC strobe implementation **Cause of error** The data in a message are contradictory (implementation of a PLC strobe with a macro). **Error correction** - Inform your service agency

Error number	Description
220-0026	**Error message** Mid-program startup: Simulated TOOL CALL missing **Cause of error** - You executed a block scan that makes a TOOL CALL. However, in the config object CfgSimPosition, the required axis positions after the tool change were not specified. **Error correction** - Adapt the machine configuration. Assign appropriate values to the config object CfgSimPosition. - Inform your machine tool builder.
220-0027	**Error message** Contradictory data in calculation of the position to be attained **Cause of error** When returning to the contour (mid-program startup), the control found contradictory data when calculating the position to be moved to. **Error correction** - Inform your service agency.
220-0028	**Error message** OK **Cause of error** **Error correction**
220-0029	**Error message** NC program **Cause of error** **Error correction**
220-002A	**Error message** NC program changed! **Cause of error** **Error correction**
220-002B	**Error message** External tool **Cause of error** **Error correction**
220-002C	**Error message** Remaining tool life too short **Cause of error** **Error correction**

Error number	Description
220-002D	**Error message** Tool life exceeded **Cause of error** **Error correction**
220-002E	**Error message** Radius difference exists **Cause of error** **Error correction**
220-002F	**Error message** Radius R2 greater than radius R **Cause of error** **Error correction**
220-0030	**Error message** Tool not defined **Cause of error** **Error correction**
220-0031	**Error message** No fitting tool available **Cause of error** **Error correction**
220-0032	**Error message** Tool locked **Cause of error** **Error correction**
220-0033	**Error message** Warning: Tool usage file was not generated with %s! **Cause of error** **Error correction**

Error number	Description
220-0034	**Error message** The current kinematic configuration uses a deactivated axis! **Cause of error** In the current kinematic configuration, an axis is used that is deactivated at present. When an NC program is started or after a PLC strobe is executed, the control checks whether all axes of the active kinematic configuration are also active. Axis movements are no longer allowed. **Error correction** - Activate the deactivated axis, check the machine configuration and correct it if required. - Activate another machine kinematic configuration through the NC program. - Edit the machine configuration or activate another machine kinematic configuration.
220-0035	**Error message** Not all axes in necessary nominal position **Cause of error** You tried to resume the program after returning to the contour, after an NC stop, or after a mid-program startup although not all axes are on the nominal position. The nominal position after an NC stop is the stop position. The nominal position after a mid-program startup is the calculated restore position. **Error correction** - Check the configuration, CfgChannelAxes/restoreAxis - Inform your machine tool builder.
220-0036	**Error message** Incorrect operating mode for internal cycle **Cause of error** An internal cycle is running in another operating mode than intended. Therefore the internal data of the control are possibly inconsistent. **Error correction** - Shut down the control and restart. - Inform your service agency.
220-0037	**Error message** Cancel during switch between turning op. and milling operation **Cause of error** A cancelation occurred during switchover between turning and milling operation. **Error correction** Run FUNCTION MODE MILL or FUNCTION MODE TURN again for consistency.

Error number	Description
220-0038	**Error message** Machine not initialized **Cause of error** - After the machine traversed the reference points, you tried to select a program run mode although the machine was not yet completely initialized. - You canceled the initialization process. **Error correction** - Close all open protective doors. - Unlock all emergency stop buttons. Then press the INIT MACHINE soft key (2nd soft-key row).
220-0039	**Error message** Data reset due to reconfiguration of the kinematics **Cause of error** Data were set during a reconfiguration of the kinematics that are independent from the kinematics. Press the "INTERNAL INFO" soft key for more information. **Error correction** - Delete the error and do an NC start if the reset is OK. - Cancel if difficulties are expected in program continuation due to the reset.
220-003A	**Error message** Data record already locked **Cause of error** An attempt was made to update the tool life in an locked tool data record. **Error correction** Cancel the lock on the data record (e.g. exit the input with the "EDIT OFF/ON" soft key), otherwise it can result in data loss at program end when the tool life updated.
220-003B	**Error message** Cannot end the cancel system cycle %1 **Cause of error** Could not complete the cancel system cycle, perhaps because a PLC strobe was not acknowledged. **Error correction** - Shut down and restart the control (shutdown via error window, MORE FUNCTIONS soft key) - Inform your machine tool builder. He should take the following measures: - Correct the error in the cancel cycle or in the OEM cancel macro - Correct the error in the PLC program

Error number	Description
220-003C	**Error message** Faulty configuration **Cause of error** Entry appears twice in the list **Error correction** Check the configuration data and edit them if necessary
220-003D	**Error message** Tool usage time could not be calculated **Cause of error** - An error occurred in finding the application times of the tools. - Tool usage file is not available or not up to date. **Error correction** - Ensure that the tool usage test has been activated by the configuration. - Usage file for NC program: simulate the program in the Test Run operating mode. The TNC then automatically creates the tool usage file. - Usage file for pallet file: simulate the marked program in the Test Run operating mode. The TNC then automatically creates the tool usage file for each simulated program.
220-003F	**Error message** Program cannot be continued. Selection with GOTO necessary. **Cause of error** You have called for the program simulation to continue within an NC block. Changed conditions must be applied in order to continue. These can be, for example, a new position for a stop, a changed Q parameter, or a changed condition for the activation of skipped blocks. **Error correction** Starting with RESET+START is possible, as is START after GOTO. As an alternative, perform the stated changes only at a stop at the beginning of an NC block.
220-0040	**Error message** File path %1 missing in CfgConfigDataFiles or in CfgJhConfigDataFiles %2 has no effect **Cause of error** A file path is missing in the configuration data. See the error text for more information. **Error correction** Enter the missing path in CfgConfigDataFiles or in CfgJhConfigDataFiles

Error number	Description
220-0041	**Error message** Programmed variable cannot be applied **Cause of error** You tried to edit a variable (e.g. a Q parameter) even though this is not possible in the current state. For example, variables of an NC program cannot be edited while the program is running (not stopped). **Error correction** Try again under appropriate conditions.
220-0042	**Error message** Warnings are being suppressed **Cause of error** The current program run is generating many warnings. The number of warnings of the same type is limited. Further warnings of this type will be suppressed. **Error correction** Correct the NC program
221-0004	**Error message** Error in the kinematic configuration: %1 **Cause of error** List attribute not initialized **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0005	**Error message** Error in module configuration **Cause of error** The object requested by the configuration server could not be found. **Error correction** - Edit the configuration data - Inform your service agency
221-0007	**Error message** Configuration error occured **Cause of error** General error message that shows that at least one configuration error has occurred. **Error correction** - Correct the displayed configuration error - If no more configuration errors occur, the message is automatically deleted.

Error number	Description
221-0008	**Error message** Error in module configuration: %1 **Cause of error** - The individual object received by the configuration server is incorrect. **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0009	**Error message** Error in module configuration: %1 **Cause of error** - The view object received by the configuration server is incorrect. **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-000A	**Error message** Error in module configuration: %1 **Cause of error** Inconsistent configuration data **Error correction** - Edit the configuration data. - Inform your service agency.
221-000B	**Error message** General system error in the geometry chain **Cause of error** Contradictory data **Error correction** Inform your service agency.
221-000C	**Error message** Incorrect condition in switch statement **Cause of error** System error during reconfiguration **Error correction** - Edit the kinematic configuration. - Inform your service agency.

Error number	Description
221-000D	**Error message** Error in module configuration: %1 **Cause of error** Uninitialized list in configuration object **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-000E	**Error message** Error in module configuration: %1 **Cause of error** List size of an attributre in configuration object is too small. **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-000F	**Error message** Kinematics configuration faulty %1 **Cause of error** Incorrect axis key in key list **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0010	**Error message** Kinematics configuration faulty **Cause of error** Error in the kinematic configuration **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0011	**Error message** Kinematics configuration faulty **Cause of error** Error in the kinematic configuration **Error correction** - Change the kinematic configuration. - Inform your service agency.

Error number	Description
221-0012	**Error message** Kinematics configuration faulty **Cause of error** Error in the kinematic configuration **Error correction** - Edit the kinematic configuration. - Inform your service agency.
221-0013	**Error message** Kinematics configuration faulty **Cause of error** Error in the kinematic configuration **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0014	**Error message** Kinematics configuration faulty %1 **Cause of error** Missing attribute **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0015	**Error message** Kinematics configuration faulty %1 **Cause of error** Attribute with incorrect value **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0016	**Error message** Kinematics configuration faulty %1 **Cause of error** Inconsistency in number of axes **Error correction** - Change the kinematic configuration. - Inform your service agency.

Error number	Description
221-0017	**Error message** Kinematics configuration faulty %1 **Cause of error** Inconsistent key lists in the kinematic model **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0018	**Error message** Kinematics configuration faulty %1 **Cause of error** Keys in key list already defined **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0019	**Error message** Kinematics configuration faulty %1 **Cause of error** Same key in the key lists for coordinate transformation through directions and coordinate transformation through angle. **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-001A	**Error message** Kinematics configuration faulty %1 **Cause of error** Incorrect index for key list **Error correction** - Change the kinematic configuration. - Inform you service agency.
221-001B	**Error message** Kinematics configuration faulty %1 **Cause of error** Incorrect number of coordinate transformations **Error correction** - Change the kinematic configuration. - Inform your service agency.

Error number	Description
221-001C	**Error message** Kinematics configuration faulty %1 **Cause of error** Missing coordinate transformation matrix **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-001D	**Error message** Kinematics configuration faulty %1 **Cause of error** Key missing in two key lists, although it should be in one of them **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-001E	**Error message** Error in the kinematic configuration Function not yet implemented: %1 **Cause of error** Attempt to use a nonimplemented function **Error correction** - Edit the NC program.
221-001F	**Error message** Kinematics configuration faulty %1 **Cause of error** Coordinate system defined in two different ways **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0020	**Error message** Kinematics configuration faulty %1 **Cause of error** - Missing coordinate system definition by angle - Usually caused by an incorrect key in a coordinate transformation defined by directions **Error correction** - Change the kinematic configuration. - Inform your service agency.

Error number	Description
221-0021	**Error message** Kinematics configuration faulty %1 **Cause of error** Incorrect key in key list **Error correction** - Change the kinematic configuration. - Inform your service agency.
221-0022	**Error message** System error during reconfiguration of geometry chain: %1 **Cause of error** System error during reconfiguration **Error correction** - Inform your service agency.
221-0023	**Error message** Incorrect path parameters for look-ahead: %1 **Cause of error** Incorrect path parameter for look-ahead **Error correction** - Edit the configuration.
221-0024	**Error message** No acceleration was defined **Cause of error** No axis acceleration set **Error correction** Edit the configuration.
221-0025	**Error message** Invalid max. feed override **Cause of error** Invalid maximum feed rate override **Error correction** Edit the configuration

Error number	Description
221-0026	**Error message** Error in the general parameter configuration: %1 **Cause of error** Error in the general parameter configuration. **Error correction** - Edit/complete the parameter configuration. - Inform your service agency.
221-0027	**Error message** Kinematics configuration faulty %1 **Cause of error** A programmable axis with special properties was configured. This programmable axis was not assigned to any physical. **Error correction** – Check the axis configuration and edit/add to it if required. - Inform your service agency
221-0028	**Error message** Bad attribute info of %1 **Cause of error** Attribute information invalid or illegible **Error correction** Inform your service agency.
221-0029	**Error message** Value too small in %1-%2 **Cause of error** Programmed or configured value too small. **Error correction** - Edit the configuration. - Edit the program.
221-002A	**Error message** Value too large in %1-%2 **Cause of error** Programmed or configured value too large. **Error correction** - Edit the configuration. - Edit the program.

Error number	Description
221-002B	**Error message** No SQL column description for column %1 in table %2 **Cause of error** The SQL server does not provide a column description for the given SQL table column. It could be that the corresponding table does not exist, is incorrect in syntax, or for some other reason cannot be opened from the SQL server. Or the table has no column with the given name. **Error correction** Ensure that the corresponding table exists and that it has the matching column. Then restart the control. The interpreter needs column descriptions - For all table columns for which you have configured bonds (over CfgSqlProperties, CfgTableBinding and CfgColumn-Binding). - For all columns of some fundamental SQL tables, that are indispensable for the correct function of the system (e.g. tool table).
221-002C	**Error message** SQL column description for column %1 in table %2 inconsistent with bond **Cause of error** The column description for the specified SQL table column provided by the SQL server does not have the correct format for the bond that you have configured for this column, or the table column has a format the interpreter does not recog-nize. **Error correction** Ensure that the column description is correct.
221-002D	**Error message** CfgTableBinding with key %1 inconsistent **Cause of error** In a CfgTableBinding you listed a key for a CfgColumnBind-ing that does not exist. **Error correction** Add the missing CfgColumnBinding or delete the entry from CfgTableBinding.

Error number	Description
221-002E	**Error message** Faulty SQL column bond for column %1 **Cause of error** A faulty bond is configured for the specified column (CfgColumnBinding). **Error correction** Correct CfgColumnBinding: You can configure bonds on Q parameters (ID=0, NR0 bis 999) and bonds on the system data managed in the interpreter.
221-002F	**Error message** Inconsistent configuration for implicit SQL access in the interpreter **Cause of error** The configuration data that determine how the interpreter implicitly accesses SQL tables are inconsistent. (These configuration data are accessible only to the control manufacturer) **Error correction** Correct the configuration data: - The tables (id50Table etc.) given in CfgChannelSysData for the corresponding channel must exist. - An entity CfgSysDataTable with the corresponding key must exist for the columns (id50Columns etc.). - For every entry in the Attribute column of the entity CfgSysDataTable, an entity CfgSysDataColumn with the corresponding key must exist.
221-0031	**Error message** Machine base not specified **Cause of error** The kinematics contain a plane (CfgCMOPlane) and rotary axes. The position of the machine base must be entered in this kinematic configuration. **Error correction** Specify the position of the machine base (in CfgKinAnchor). Use only planes between the machine base and the first rotary axis at the tool. Use only planes between the machine base and the first rotary axis at the machine table.
221-0032	**Error message** Kinematics configuration faulty **Cause of error** Physical axis cannot be assigned to any programmable axis **Error correction** - Change the configuration (CfgProgAxis, CfgAxis) - Inform your service agency

Error number	Description
221-0033	**Error message** Model not loaded "%1": CMOMesh3D "%2" **Cause of error** The M3D-file %1 could not be opened. **Error correction** Check the path to the file in the machine configuration and correct it if necessary.
221-0034	**Error message** Model not loaded "%1": CMOMesh3D "%2" **Cause of error** Error during read-in of the M3D file. The file is corrupted or not a valid M3D file. **Error correction** - Check the file path and correct it if required. - Re-install the file
221-0035	**Error message** STL model does not fulfill the quality requirements **Cause of error** STL model %1 does not fulfill the quality requirements. **Error correction** Use an STL model that fulfills the quality requirements. The following requirements are placed on STL models: - All size values in mm - No gaps between triangles ("waterproof") - No overlapping - No degenerated triangles Refer to the additional information in the Technical Manual.
221-0036	**Error message** Kinematic temperature compensation incorrectly configured **Cause of error** Parameters in the machine configuration were entered incorrectly: Within the config object CfgKinSimpleTrans, both machine parameters realtimeComp and temperatureComp are set. This is not allowed. Only one of the two parameters can be set. **Error correction** Correct the machine configuration: Delete either the parameter realtimeComp or temperatureComp.

Error number	Description
221-0037	**Error message** Model not loaded "%1": CMOMesh3D "%2" **Cause of error** Error while loading the M3D file. The file contains too many triangles. **Error correction** Model the collision objects with fewer triangles. Refer to the additional information in the Technical Manual.
221-006F	**Error message** Faulty kinematics configuration Active kinematics %1 contains an invalid insertion point. **Cause of error** The given kinematic model contains at least one invalid insertion point for a tool-carrier kinematic model (entry under CfgKinToolSocket) **Error correction** Ensure that the kinematic model contains no more than one object of the CfgKinToolSocket type. Ensure that no objects of the CfgKinSimpleAxis or CfgKinAnchor types are located between the tool (i.e. the upper end of the kinematic chain) and the insertion point for the tool carrier.
221-0071	**Error message** No facing slide axis available in the kinematic model **Cause of error** The kinematics do not include a facing slide axis. **Error correction** - Change the kinematic configuration - Inform your service agency
221-0072	**Error message** Spindle or facing slide in faulty in the kinematic model **Cause of error** The spindle is not correctly configured in the kinematics: - In the kinematics configuration, the spindle is not located directly next to the facing slide - The kinematics do not include spindle **Error correction** - Inform your service agency - Adapt the kinematic configuration

Error number	Description
221-0073	**Error message** Datum of facing slide axis is incorrect **Cause of error** The datum of the facing slide is not on the spindle axis. **Error correction** - Check the kinematic configuration and adapt it if necessary. - Inform your service agency.
221-0074	**Error message** Faulty kinematics configuration **Cause of error** Active kinematics contains an invalid insertion point for a fixture (entry under CfgKinFixSocket). The invalid entry is shown in the additional information of the error message. **Error correction** Ensure that the kinematic model contains no more than one object of the CfgKinFixSocket type. Ensure that no objects of the type CfgKinSimpleAxis or CfgKinAnchor are located between the machine table (i.e. the bottom end of the kinematics chain) and the insertion point for the fixture.
221-0075	**Error message** Faulty kinematics configuration **Cause of error** Active kinematics contains an invalid insertion point for a tool-carrier kinematic model (entry under CfgKinToolSocket). The invalid entry is shown in the additional information of the error message. **Error correction** Ensure that the kinematic model contains no more than one object of the CfgKinToolSocket type. Ensure that no objects of the type CfgKinSimpleAxis or CfgKinAnchor are located between the tool (i.e. the top end of the kinematics chain) and the insertion point for the tool carrier.
230-0001	**Error message** Parameter set %2 of axis %3 does not exist **Cause of error** An undefined axis parameter block was selected. **Error correction** In the configuration data, create an additional parameter block for this axis, or select another parameter block for this axis.

Error number	Description
230-0002	**Error message** Logical axis number %2 too large **Cause of error** The control supports a certain maximum number of axes. Here more axes were configured that permitted. **Error correction** Configure fewer axes.
230-0003	**Error message** Too many analog axes configured (more than 2) **Cause of error** The control supports a certain maximum number of analog axes. Here more axes were configured than allowed. **Error correction** Configure fewer analog axes.
230-0004	**Error message** More axes are activated than are enabled in the SIK **Cause of error** The axis options in the SIK specify how many axes can be active at the same time. You have activated more axes over the machine configuration or the PLC program than are allowed as axis options in the SIK. You can delete this error message. If the configured axis number is still too large after the drives are switched on again, the error message will reappear. **Error correction** - Check the machine configuration and PLC program. - If you need more axes, you can get a code number from HEIDENHAIN to enable them.
230-0005	**Error message** External EMERGENCY STOP External EMERGENCY STOP **Cause of error** - The PLC input for the control-is-ready signal is inactive - The EMERGENCY STOP circuit was interrupted manually or by the control. **Error correction** - Enable the EMERGENCY STOP button, switch on the control voltage, and acknowledge the error message. - Check the EMERGENCY-STOP circuit. (EMERGENCY STOP button, axis limit switches, wiring, etc.)

Error number	Description
230-0006	**Error message** Check the parameter for the direction of spindle rotation (2)! **Cause of error** A change in the evaluation of the parameter signCorrNominalVal automatically changed the value of the parameter signCorrActualVal. **Error correction** Please check whether the spindle turns with M3 and M19 in the correct direction. If required, use parameter CfgAxisHardware > signCorrNominalVal or CfgAxisHardware > signCorrActualVal to define the direction of rotation correctly according to the data in the Technical Manual.
230-0007	**Error message** Channel number %2 too large **Cause of error** The control supports a certain maximum number of channels. Here more channels were configured than allowed. **Error correction** Configure fewer channels.
230-0008	**Error message** No reaction from CC **Cause of error** The speed and current controller was switched off due to an error. **Error correction** Check the axis cabling.
230-0009	**Error message** IPO exceeds cycle time **Cause of error** The control loop exceeds the maximum permissible cycle time. **Error correction** Increase the maximum permissible cycle time in the System->MachineHardware->ipoCycle parameter.
230-000A	**Error message** Axis %2 is switched inactive **Cause of error** Command to an axis configured as inactive. **Error correction** In the parameter "Axes->PhysicalAxes->????->axisMode," switch the axis to "active." "????" designates the current axis name.

Error number	Description
230-000B	**Error message** This is not the export version of the software **Cause of error** This is not the export version of the software. **Error correction** Inform your service agency.
230-000C	**Error message** Parameter %2 is not loaded until the control has been restarted **Cause of error** A parameter cannot be loaded for this axis without a RESET of the control. **Error correction** Restart the control.
230-000D	**Error message** Ipo-Trace started **Cause of error** IPO trace started (info) **Error correction**
230-000E	**Error message** Ipo-Trace stopped **Cause of error** IPO trace stopped (info) **Error correction**
230-000F	**Error message** Drive switched off illegally %2 **Cause of error** The drive was switched off without a command from the PLC. **Error correction**
230-0010	**Error message** IPO is running in simulation mode **Cause of error** IPO is running in simulation mode (info) **Error correction**

Error number	Description
230-0011	**Error message** This software version has not been enabled **Cause of error** Wrong software installed **Error correction** Inform your service agency
230-0012	**Error message** Only %1 KB of free memory remaining in SYS partition **Cause of error** The memory capacity of the SYS partition is almost depleted. **Error correction** Inform your service agency
230-0013	**Error message** Only %1 KB of free memory remaining in SYS partition **Cause of error** The memory capacity of the SYS partition is almost depleted. Service information recording was stopped. **Error correction** Inform your service agency
230-0014	**Error message** Parameter selection for axis %2 not allowed in this state **Cause of error** A parameter block switchover was requested in an illegal condition of the NC. **Error correction** Check the PLC program
230-0015	**Error message** Initialization of counter components (G50) failed **Cause of error** The required configuration data for the counter component (G50) could not be read from the file %SYS%\config\CfgG50Init.cfg. **Error correction** Inform your service agency

Error number	Description
230-0016	**Error message** Wrong hardware configuration **Cause of error** There are two SPI modules with various versions. **Error correction** Inform your service agency
230-0017	**Error message** Access to internal periphery failed **Cause of error** A timeout was caused during access to the internal periphery **Error correction** Inform your service agency
230-0018	**Error message** Position or speed control of axis %2 still active **Cause of error** The position, speed and current controllers of axes must be switched off before they can be activated or deactivated. **Error correction** - Check the PLC program
230-0019	**Error message** Changing a parameter requires an NC STOP **Cause of error** During reconfiguration or a parameter set switchover a parameter was changed that requires a previous NC STOP. **Error correction** - Check the PLC program
230-001A	**Error message** A parameter change requires the drive to be switched off (axis %2) **Cause of error** During reconfiguration or a parameter set switchover a parameter was changed that requires that the drive be switched off beforehand. **Error correction** - Check the PLC program

Error number	Description
230-001B	**Error message** For a parameter change the drive has to be deactivated (axis %2) **Cause of error** During reconfiguration or a parameter set switchover a parameter was changed that requires that the drive be deactivated beforehand. Note: The changed parameter or parameter set was not accepted. **Error correction** - Check the PLC program
230-001C	**Error message** Interrupt cycle greater than 3 ms **Cause of error** The cycle time of the controller interrupt exceeds the maximum permissible tolerance of 3 ms. The cause could be a hardware defect of the computer unit MC. **Error correction** - Inform your service agency
230-001D	**Error message** Control loop of axis %2 was opened **Cause of error** The position control loop was opened in order to optimize the axis (e.g. with TNCopt). **Error correction**
230-001E	**Error message** Timeout in the initial servicing of axis %2 **Cause of error** Possible cause: There is no connection to the PC initial-servicing software TNCopt. **Error correction** - Check the connection to TNCopt. (Is the network cable plugged in? Are the interface settings correct?) - Restart TNCopt
230-001F	**Error message** Max. traverse range limits of axis %2 exceeded **Cause of error** When the control loop was open the traverse range limits given by TNCopt were exceeded. **Error correction** Inform your service agency

Error number	Description
230-0020	**Error message** Failed to send internal message **Cause of error** Error in the internal system communication **Error correction** Inform your service agency
230-0021	**Error message** Axis %2 cannot be activated **Cause of error** The value configured under CfgAxis-axisHw prohibits this activation command. **Error correction** Check the configuration
230-0022	**Error message** One or more axes of the channel (%2) are deactivated **Cause of error** You have selected a machine kinematic configuration that contains deactivated axes. During NC start the control checks whether all axes of the selected kinematic configuration are also active. **Error correction** - Check the machine configuration and PLC program. - Activate deactivated axes. - Select a machine kinematic configuration that has no deactivated axes.
230-0023	**Error message** Change of the activation status of axis %2 not allowed **Cause of error** A change was requested of the activation status of an axis (activate/deactivate) in an illegal status of the NC. **Error correction** - Check the PLC program and correct if necessary.
230-0024	**Error message** SPI analog module on CC%2 not recognized **Cause of error** An analog axis was configured on a CC, but no SPI analog module was detected there. **Error correction** Check the configuration. If necessary, inform your service agency.

Error number	Description
230-0025	**Error message** Position of axis %2 stored **Cause of error** The position of this axis is to be saved (frozen) while the control loop is closed or the control loop of this axis was to be closed while the position was saved. **Error correction** Check the PLC program. If necessary, inform your service agency.
230-0026	**Error message** An HSCI participant has triggered the SS2/STOP2 stop reaction **Cause of error** - The self-test of the control could not be completed - A device reports a temperature problem - The fan of a device is defective **Error correction** - Note further messages - Use the bus diagnostics (or TNCdiag, if available) to find out which device reports the error (bit REQ.SS2 in the local S status)
230-0027	**Error message** More spindles than permitted are configured **Cause of error** You configured more spindles than allowed for the control. **Error correction** - Check the machine configuration and correct it if required Parameter: System / CfgAxes / spindleIndices
230-0029	**Error message** IPO cycle time exceeded permissible threshold (%2 us) **Cause of error** The cycle time of the controller interrupt exceeds the default threshold in the internal parameter maxIpoTime. **Error correction** Inform your service agency.

Error number	Description
230-002A	**Error message** Difference between ACTL and NOML spindle speed (%2) too high **Cause of error** The difference between the actual and nominal speed exceeds the tolerance. **Error correction** - Check the parameters CfgSpindle/absSpeedTolerance and CfgSpindle/relSpeedTolerance - Check whether the actual and nominal speeds have the same algebraic sign
230-002B	**Error message** Formula entered in distPerMotorTurnF is invalid **Cause of error** In the machine parameter "distPerMotorTurnF" you entered a formula that contains invalid characters. **Error correction** - Check the input value in machine parameter "distPerMotor-TurnF " and correct it
230-002C	**Error message** PLC:/ccfiles file must be deleted **Cause of error** The file PLC:/ccfiles is available, but it is needed as a directory by the NC software for CC files. **Error correction** Delete the PLC:/ccfiles file and then restart the control
230-002D	**Error message** Deactivation of an active touch probe (TS or TT) not allowed **Cause of error** The PLC program tried to deactivate a touch probe activated by the NC, or the NC tried to deactivate a touch probe activated by the PLC. **Error correction** Check the NC program and/or the PLC program
230-002E	**Error message** Initialization of counter components (G127) failed **Cause of error** Hardware is defective **Error correction** Inform your service agency.

Error number	Description
230-002F	**Error message** The control is still in its factory default setting **Cause of error** The parameter CfgMachineSimul/simMode is still set to the value "Delivery". The drives cannot be switched on in this mode. **Error correction** - Set parameter CfgMachineSimul/simMode to the value "FullOperation". First the parameters of the axes must be assigned with realistic values.
230-0030	**Error message** Encoder error in axis %2 **Cause of error** The encoder for this axis reports an error. Possible errors (encoder status): Bit 2 = 1: Position could not be ascertained Bit 3 = 1: CRC error during EnDat 2.2 transmission Bit 4 = 1: No position measurement with EnDat 2.2 Bit 5 = 1: Alarm 1 with EnDat 2.2 Bit 6 = 1: Alarm 2 with EnDat 2.2 Bit 7 = 1: Timeout during EnDat 2.2 transmission **Error correction** Check the connected encoder
230-0031	**Error message** RTC: Axis %2 exceeds the max. permissible velocity **Cause of error** The maximum permissible value was exceeded during the real-time coupling (RTC). **Error correction** Allow a greater share of CfgFeedLimits/maxFeed in CfgRtCoupling/maxFeed, or change the function in CfgRtCoupling/function
230-0032	**Error message** RTC: Axis %2 exceeds the max. permissible acceleration **Cause of error** The real-time coupling function (RTC) causes a violation of the maximum permissible acceleration. **Error correction** Allow a greater share of CfgFeedLimits/maxAcceleration in CfgRtCoupling/maxAcc, or change the function in CfgRtCoupling/function

Error number	Description
230-0033	**Error message** RTC: Axis %2 exceeds the max. permissible end positions **Cause of error** The real-time coupling function (RTC) causes a violation of the maximum permissible working space. **Error correction** Adjust the setting in CfgRtCoupling/function
230-0034	**Error message** RTC: Axis %2 causes a run-time error **Cause of error** The configured function for Real-Time Coupling (RTC) causes a run-time error (e.g. root(-1)). **Error correction** - Check the function in the machine parameter CfgRTCou-pling/function and adapt it if required - Inform your service agency
230-0034	**Error message** The formula in limitAccSpeedCtrlF is faulty **Cause of error** In the machine parameter "distPerMotorTurnF", you entered a formula that contains invalid characters. **Error correction** Check the input value in the parameter "distPerMotorTurnF" and correct it
230-0035	**Error message** The formula in limitDecSpeedCtrlF is faulty **Cause of error** In the machine parameter "limitDecSpeedCtrlF", you entered a formula that contains invalid characters. **Error correction** Check the input value in the parameter "limitDecSpeedCtrlF" and correct it

Error number	Description
230-0036	**Error message** Voltage drop on controller unit **Cause of error** The power supplies on a device in the HSCI line are outside of the specified range. The HSCI bus diagnosis indicates which HSCI component triggered the error. Possible devices: - MC main computer - PL inputs/outputs - MB machine operating panel - Other CC in the HSCI line Possible causes: - Insufficient power supply to the devices - Short circuit in the power supply - Short circuit in PL inputs and outputs **Error correction** - Check the supply voltage in the connected devices Check the wiring for possible short circuits (e.g. PLC inputs or outputs). - If required, exchange the defective hardware - Inform your service agency
230-0037	**Error message** Impermissibly large position nominal value in axis %2 **Cause of error** - Internal software error - Faulty nominal value jump detected **Error correction** - Save the service files - Inform your service agency
230-003A	**Error message** Measuring cycle started without a touch probe **Cause of error** - Touch probe cycle started for measurement without inserted touch probe **Error correction** - Check the NC program - Insert the touch probe - Inform your service agency

Error number	Description
230-003B	**Error message** Parameter checking: %2 warnings issued **Cause of error** - Current configuration of the machine has discrepancies **Error correction** - Check the inspection results in PLC:\service\Param-Check.txt - Correct any discrepancies - Inform your service agency
230-003D	**Error message** Two-fold positioning was requested for axis %2 **Cause of error** - Double positioning started for an axis - The axis is to be moved both by the PLC as well as the NC **Error correction** - Check the PLC program and correct it if necessary - Inform your service agency
230-003F	**Error message** Position encoder input reserved by FS (axis %2) **Cause of error** In systems with functional safety, the speed and position encoder inputs are always permanently assigned to a single axis. This means in a single-encoder system (speed encoder only) it is not possible for example to use the vacant position encoder input for another drive or a display axis. **Error correction** - Check the configuration and change it if necessary - Inform your service agency.
230-0040	**Error message** Position encoder input reserved **Cause of error** In systems with functional safety, the speed and position encoder inputs are always permanently assigned to a single axis. This means in a single-encoder system it is not possible for example to use the vacant position encoder input for another drive or a handwheel. **Error correction** - Check the configuration and change it if necessary - Inform your service agency.

Error number	Description
230-0041	**Error message** IPO cycle time has exceeded the permissible threshold (%2 µs) **Cause of error** - Internal error: Cycle time of the controller interrupt is too large. **Error correction** - Make a service file - Inform your service agency
230-0042	**Error message** The touch probe monitor is deactivated for %2 seconds **Cause of error** The probe monitoring has been deactivated by the operator for a certain time **Error correction** Retract the touch probe and/or remove it from the working space
230-0043	**Error message** Error in reference point acquisition in axis %2 **Cause of error** An error occurred during determination of the EnDat switch-on position **Error correction** Inform your service agency
230-0044	**Error message** Error in axis simulation **Cause of error** Incorrect call of the controller unit during the simulation. **Error correction** Inform your service agency.
230-0045	**Error message** Error in CfgAnalogSync (key = %2) **Cause of error** No function was configured in an element. **Error correction** - Check the configuration and correct it if necessary - Inform your service agency.

Error number	Description
230-0046	**Error message** Error in CfgAnalogSync **Cause of error** No free list element was found. **Error correction** - Check the configuration and correct it if necessary - Inform your service agency.
230-0048	**Error message** The axis (%2) cannot be traversed additionally **Cause of error** Due to the active kinematics model, this axis cannot be traversed additionally. Possible causes: - Basic rotation activated - TCPM activated - Machine with oblique axis **Error correction** Deactivate basic rotation Deactivate TCPM
230-0049	**Error message** Timeout during job acknowledgment **Cause of error** The module that assigned commands to the CC (UVR commands) can't be reached. **Error correction** Inform your service agency
230-004A	**Error message** Could not send message to PLC **Cause of error** An acknowledgment message could not be sent to the PLC. The incoming queue of the PLC is full. **Error correction** Inform your service agency
230-0064	**Error message** Axis %2 in channel %3 not yet configured **Cause of error** An axis unknown to the system is to be moved **Error correction** Check the NC program, and if required, configure the axes.

Error number	Description
230-0065	**Error message** Grinding function not available **Cause of error** Grinding function was not enabled. **Error correction** Check the NC program. If necesary, configure the grinding axes.
230-0066	**Error message** Internal error in grinding functions **Cause of error** Internal error in the grinding generators for swinging and infeed **Error correction** Inform your service agency
230-0067	**Error message** Grinding command not allowed in this state **Cause of error** Command not permitted in current state of the grinding generators **Error correction** - Check sequence of grinding commands. - Inform your service agency if necessary.
230-006C	**Error message** Stylus deflected in %2 outside of the probing process **Cause of error** The touch probe was triggered although the measuring procedure had not yet begun. **Error correction** Check the NC program or working space.
230-006D	**Error message** In %2 no axis polynomial available during movement **Cause of error** Timing problem in the coordination of interpolator and LookAhead. **Error correction** Inform your service agency.

Error number	Description
230-006E	**Error message** %2 Parameter CfgLiftOff off **Cause of error** In the parameter NcChannel->????->CfgLiftOff->on, the lift-off is switched off with NC stop although it should be activated in the NC program. "????" stands for the current channel name. **Error correction** Check the NC program or activate CfgLiftOff
230-006F	**Error message** %2 Distance CfgLiftOff != Parameter **Cause of error** The lift-off height in the NC program is greater than that entered in the parameter NcChannel->????->CfgLiftOff->distance. "????" stands for the current channel name. **Error correction** Change the lift-off distance in the NC program.
230-0070	**Error message** Too many axes to be interpolated **Cause of error** The maximum allowed number of simultaneously moving axes was exceeded. (In the export version the maximum is 4 axes.) **Error correction** Check the NC program
230-0071	**Error message** Spindle is not yet referenced **Cause of error** An spindle without reference is supposed to be positioned. **Error correction** - Check the NC program - Home the spindle
230-0072	**Error message** Too many labels in channel %2 during channel synchronization **Cause of error** Too many labels assigned during channel synchronization. **Error correction** - Check the NC program

Error number	Description
230-0073	**Error message** Error in synchronization on coordinates in channel %2 **Cause of error** A channel that is supposed to be waited for before starting channel synchronization has already traversed the next synchronization mark; i.e. the synchronization is faulty. **Error correction** - Check the NC program
230-0074	**Error message** Thread with incorrect spindle **Cause of error** You tried to drill/cut threads with a spindle that does not at present belong to this channel. **Error correction** - Check the NC program
230-0075	**Error message** This function is permitted only for modulo axes (axis %2) **Cause of error** An axis was being placed in modulo limits although it was not defined as a modulo axis in the machine configuration. Only a modulo axis can be placed in modulo limits. **Error correction** - Check the NC program - Check the machine parameter CfgAxis->isModulo and correct it if required.
230-0076	**Error message** No axes-movement allowed in channel %2 **Cause of error** The NC program was not started with the the NC Start key, so no axis movements are allowed. Or, one or more unreferenced axes are to be moved in one cycle. **Error correction** - Check the NC program - Move the axes over the reference marks
230-0077	**Error message** Illegal jump in the path profiles of an axis **Cause of error** The actual position of an axis does not agree with the nominal value calculated from the geometry. **Error correction** - Inform your service agency

Error number	Description
230-0078	**Error message** Spindle synchronization is not possible! **Cause of error** The programmed starting length for synchronizing the spindle is insufficient. **Error correction** - Increase the starting length or reduce the spindle speed.
230-0079	**Error message** Spindle not synchronized at beginning/end of the thread! **Cause of error** The programmed starting length/overtravel length for synchronizing and desynchronizing the spindle is insufficient. As a result, the thread at the start/end does not have the programmed pitch! **Error correction** - Increase the starting length/overtravel length, or reduce the spindle speed.
230-007A	**Error message** One or more axes do not reach the control window in channel %2 **Cause of error** At program start and during an exact stop, all axes must be in the control window.One or more axes of this channel has not fulfilled this condition. **Error correction** Check the configuration data CfgControllerTol->posTolerance and CfgControllerTol->timePosOK. Adapt the configuration data to the conditions of the machine.
230-007B	**Error message** One or more axes in channel %2 will be assigned by the PLC **Cause of error** For NC stop, the PLC must cancel all PLC positioning of this channel's axes. It has up to 10 seconds for this task. This time was exceeded. **Error correction** Check the PLC program
230-007C	**Error message** IPO internal Breakpoint reached **Cause of error** **Error correction**

Error number	Description
230-007D	**Error message** Thread cutting canceled with NC stop **Cause of error** The NC stop button was pressed in channel %2 during thread cutting **Error correction** Restart NC program
230-007E	**Error message** Axis %2 in channel %3 has no reference **Cause of error** One axis of the active kinematic configuration has no reference. **Error correction** Reference the axis and restart the NC program.
230-007F	**Error message** Retraction from thread completed **Cause of error** The retraction from the thread has been concluded. **Error correction** NC program cannot be continued. If required, restart the program.
230-0080	**Error message** Auxiliary axes are not permitted in an NC channel **Cause of error** Free auxiliary axes (e.g. from a UMC 11x), are supposed be adopted in the kinematics of an NC channel. This is not allowed. **Error correction** Check the machine configuration and correct it if required.
230-0081	**Error message** Finding the field angle **Cause of error** **Error correction**
230-0082	**Error message** Touch probe cannot be switched off **Cause of error** The control tried to switch off the touch probe but it did not react within a specified time. **Error correction** - Check the signal. – Check the transmitter/receiver and cleaned it if necessary.

Error number	Description
230-0083	**Error message** “MoveAfterRef” is not possible with active protection zone **Cause of error** - A protection zone was defined for a modulo axis. - A configured movement is not possible after the reference run. **Error correction** - Check the axis position - After the reference run, delete the movement in the configuration - Inform your service agency
230-0084	**Error message** Dual-head evaluation not permitted in the export software (%2) **Cause of error** - The dual-head evaluation function requires an export license. - The parameter MP_posEncoderTwoHead may not be set in the export software. **Error correction** - Check and correct the configuration. - Inform your service agency.
230-00C7	**Error message** Coupling for inactive axes not possible (axis %2) **Cause of error** A coupling for a deactivated axis was supposed to be closed. This is not possible. **Error correction** Check the PLC program or configuration. Inform your service agency.
230-00C8	**Error message** No configuration available for axis %2 **Cause of error** There is no configuration available for the desired axis coupling. **Error correction** A desired coupling (position coupling or torque coupling) for the respective slave axis must be entered under CfgAxisCoupling.

Error number	Description
230-00C9	**Error message** Axis %2 is already a master axis **Cause of error** For the desired axis coupling, the slave axis is already a master axis. **Error correction** Couplings can be commanded only for axes that are not the master or slave axis of an already existing coupling.
230-00CA	**Error message** Axis %2 is already a slave axis **Cause of error** For the desired axis coupling, the slave axis is already a slave axis. **Error correction** Couplings can be commanded only for axes that are not the master or slave axis of an already existing coupling.
230-00CB	**Error message** No coupling active (axis %2) **Cause of error** An attempt was made to open an axis coupling that is already active. **Error correction** Only active couplings can be opened.
230-00CC	**Error message** Coupling is modulo / non-modulo axes not allowed (axis %2) **Cause of error** Differently configured axes were supposed to be coupled. **Error correction** For axis coupling, either both or neither of the axes (master and slave) have to be modulo axes.
230-00CD	**Error message** Opening an axis coupling only allowed for slave axis (axis %2) **Cause of error** An axis coupling was supposed to be opened. The command for opening must be sent to the slave axis. **Error correction** Check the PLC or NC program

Error number	Description
230-00CE	**Error message** Maximum position difference exceeded (axis %2) **Cause of error** The position difference configured in parameter CfgAxisCoupling->maxPosDiff was exceeded. **Error correction** Check the machine or parameters.
230-00CF	**Error message** Maximum position difference exceeded (axis %2) **Cause of error** The position difference configured in parameter CfgAxisCoupling->ultimatePosDiff was exceeded. This error is not deletable because it is a mechanical defect. **Error correction** Check the mechanical configuration or parameters.
230-00D0	**Error message** Slave does not reach coupling position. (axis %2) **Cause of error** During coupling the slave axis crossed over its own software limit switch. **Error correction** Check the position of the axes (master and slave) and the parameters.
230-00D1	**Error message** Option for gantry axes not enabled **Cause of error** A gantry axis (synchronized axes with position coupling) was configured and activated, but the required software option was not yet enabled. **Error correction** - Check the parameter object CfgAxisCoupling - Enable the software option
230-00D2	**Error message** Coupling factor not equal to +1 or -1 is not allowed **Cause of error** For modulo axes, only coupling factors of +1 or -1 are allowed for a gantry coupling. **Error correction** Check the parameter object CfgAxisCoupling or the PLC program

Error number	Description
230-00D3	**Error message** Option for spindle synchronism not enabled **Cause of error** A spindle synchronization was commanded, but the required software option was not enabled. **Error correction** Enable the software option
230-00D4	**Error message** During active synchronism, a spindle cannot be used as an axis **Cause of error** During active synchronism, a spindle is to be used as an interpolating axis **Error correction** - Check the NC or PLC program and adapt it if necessary - Inform your service agency
230-00D5	**Error message** Kinematics axis not possible as slave axis. (axis %2) **Cause of error** An axis that is in the kinematics cannot be used as a slave axis of a gantry system. **Error correction** - Check the axis configuration. - Check the kinematics configuration. - Inform your service agency.
230-00FA	**Error message** Limit switch %2 + Limit switch %1 + **Cause of error** The calculated tool path exceeds the machine's positive traverse limits. **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point.

Error number	Description
230-00FB	**Error message** Limit switch %2 - Limit switch %1 - **Cause of error** The calculated tool path exceeds the machine's traverse limits. **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point.
230-00FE	**Error message** Pos. SW limit switch is smaller than neg. SW limit switch (%1) **Cause of error** The value of the positive SW limit switch is smaller than the value of the negative SW limit switch **Error correction** Check the parameter Axes->ParameterSets->????->CfgPositionLimits->... "????" stands for the current parameter block name
230-00FF	**Error message** The PLC variable %1 has reached the maximum value of %2 mm **Cause of error** The variable concerned goes into the calculation of the kinematic compensation and has exceeded the maximum permissible value. The variable will be set to the maximum value. The warning will be deleted as soon as the variable has fallen below the maximum value by 0.1 mm. **Error correction** - Check the calculation for the variable's value - Inform your machine tool builder
230-0100	**Error message** %1 axis has reached the max. axis-error compensation of %2 mm **Cause of error** The calculated axis-error compensation has exceeded the max. permissible value for the axis. The compensation is set to the maximum value. The warning is deleted as soon as the axis compensation has fallen below the maximum value by 0.1 mm. **Error correction** - Check the parameters for axis error compensation. - Check the values in the axis-error compensation table.

Error number	Description
230-0104	**Error message** In %2, the load limit 1 of %3 was exceeded **Cause of error** During load monitoring the warning threshold for power was exceeded. **Error correction** - Reduce feed rate of machining - If necessary, run a reference search to reestablish the power limits
230-0105	**Error message** In %2, the load limit 2 was exceeded by %3 **Cause of error** During load monitoring (power) the threshold for program cancelation was exceeded. **Error correction** - Reduce feed rate of machining - If necessary, run a reference search to reestablish the power limits
230-0106	**Error message** In %2 the total load limit of %3 was exceeded **Cause of error** During load monitoring the warning threshold for the total load was exceeded. **Error correction** - Reduce feed rate of machining - If necessary, run a reference search to reestablish the power limits
230-0109	**Error message** Error in the real-time coupling function (RTC) of axis %2 **Cause of error** Real-time coupling function (RTC) is supposed to be opened, but no coupling function is active. **Error correction** - Check the NC program and correct if necessary - Inform your machine tool builder
230-010A	**Error message** Error in the real-time coupling function (RTC) of axis %2 **Cause of error** Real-time coupling (RTC) is supposed to be closed, but coupling function is already active. **Error correction** - Check the NC program and correct if necessary - Inform your machine tool builder

Error number	Description
230-010B	**Error message** Error in the real-time coupling function (RTC) of axis %2 **Cause of error** No valid command for closing or opening the coupling was given for the real-time coupling function. **Error correction** - Check the NC program and correct if necessary - Inform your machine tool builder
230-010C	**Error message** Error in the real-time coupling function (RTC) of axis %2 **Cause of error** The PLC program tried to open a coupling function activated by the NC, or die NC tried to open a coupling function activated by the PLC. **Error correction** - Check the NC program and correct if necessary - Inform your machine tool builder
230-010D	**Error message** Error in the real-time coupling function (RTC) of axis %2 **Cause of error** An error occurred during compilation of the function (see INTERNAL INFO soft key). **Error correction** - Check the real-time coupling function (RTC) to be activated - Inform your service agency
230-010E	**Error message** Error in the real-time coupling function (RTC) of axis %2 **Cause of error** No function was entered in the configuration for closing a real-time coupling function (RTC). **Error correction** - Check the function under CfgRtCoupling/function and adjust if necessary - Inform your machine tool builder
230-010F	**Error message** Error in the real-time coupling function (RTC) of axis %2 **Cause of error** The active real-time coupling function (RTC) has caused a run-time error. (E.g. sqrt(-1)) **Error correction** - Check the active function in the machine configuration (CfgRtCoupling/function) - Inform your machine tool builder

Error number	Description
230-0110	**Error message** Option for coupling functions not enabled **Cause of error** A coupling was commanded, but the required software option was not enabled. **Error correction** - Enable option #135 (synchronizing functions)
230-0111	**Error message** Error in the real-time coupling function (RTC) of axis %2 **Cause of error** The function entered for closing a real-time coupling function (RTC) was too long. **Error correction** - Check the function under CfgRtCoupling/function and adjust if necessary - If the formula was presented by the PLC, check the PLC program - Inform your machine tool builder
230-0112	**Error message** RTC coupling programmed with active DCM. Deactivate DCM? **Cause of error** During active DCM collision monitoring you started a real-time coupling function (RTC). Caution: DCM must be deactivated! **Error correction** Press NC start to confirm deactivation of DCM and continue editing the program
230-0113	**Error message** RTC coupling programmed with active DCM **Cause of error** During active DCM collision monitoring you started a real-time coupling function (RTC). The NC program run was aborted. **Error correction** Adapt the NC program: Deactivate DCM if the real-time coupling function (RTC) is switched on by a cycle.

Error number	Description
230-0115	**Error message** Formula is erroneous **Cause of error** Faulty formula in the entity RTCanalog. **Error correction** - Check the configuration and change it if necessary - Inform your service agency
230-0116	**Error message** Extended limit switch monitoring %2 + **Cause of error** A compensation movement traverses the extended positive limit switch **Error correction** - Check compensations - Inform your service agency
230-0117	**Error message** Extended limit switch monitoring %2 - **Cause of error** A compensation movement traverses the extended negative limit switch **Error correction** - Check compensations - Inform your service agency
230-011A	**Error message** Run-time error in the formula calculation of offsetForM19 **Cause of error** The active formula for offsetForM19 caused a run-time error, such as sqrt(-1). **Error correction** Check the active function in the machine configuration (CfgSpindle/offsetForM19)
230-011B	**Error message** Formula in offsetForM19 invalid **Cause of error** In the machine parameter "offsetForM19", you entered a formula that contains invalid characters. **Error correction** Check the input value in the parameter "offsetForM19" and correct it

Error number	Description
230-015E	**Error message** Error in initialization of touch probe **Cause of error** 3-D touch probe: Actual position capture was refused by the CC with an error message. **Error correction** - Inform your service agency
230-0190	**Error message** Excessive servo lag in %2 **Cause of error** The following error of a moving axis is greater than the value specified in the configuration datum Axes > ParameterSets > ???? > CfgPosControl > servoLagMin1 / servoLagMax1. "????" designates the name of the affected parameter set. **Error correction** - Reduce the contouring feed rate, increase the rotational speed. - Remove any possible sources of vibration. - Inform your service agency if the error occurs frequently.
230-0192	**Error message** Excessive following error in %2 **Cause of error** The following error of a moving axis is greater than the value specified in the configuration datum Axes > ParameterSets > ???? > CfgPosControl > servoLagMin2 / servoLagMax2. "????" designates the name of the affected parameter set. **Error correction** - Reduce the contouring feed rate, increase the rotational speed. - Remove any possible sources of vibration. - Inform your service agency if the error occurs frequently.
230-0193	**Error message** Position encoder %2: Amplitude too small **Cause of error** The amplitude of the position encoder signal is too low or the signal for contamination is active. **Error correction** Check the amplitude of the position encoder signal.

Error number	Description
230-0194	**Error message** Position encoder %2: Frequency too high **Cause of error** The maximum input frequency was exceeded at a position encoder input. **Error correction** Check the input frequency of the position encoder signal.
230-0195	**Error message** Error in zero pulse distance of encoder %2 **Cause of error** Encoder defective **Error correction** Exchange the encoder.
230-0196	**Error message** Position encoder %2 defective **Cause of error** Contradiction apparent from comparison of the absolute and incremental positions. **Error correction** Inform your service agency.
230-0197	**Error message** Error in zero pulse distance of encoder %2 **Cause of error** Contradiction in comparison of the absolute and incremental position. **Error correction** Inform your service agency.
230-0198	**Error message** Nominal speed value too high %2 **Cause of error** An excessively high nominal speed value was calculated. Analog axes: Maximum nominal value +-10 V Analog spindle: Maximum nominal value +-10 V Digital axes and spindle: Maximum nominal value = maximum motor speed - The machine does not reach the set acceleration and braking ramps - Hardware error in the control loop **Error correction** - Analog axes: Check the servo - Inform your service agency

Error number	Description
230-0199	**Error message** Movement monitoring error in %2 **Cause of error** Movement monitoring: Nominal rpm=0, actual rpm=0, feed value>0 ==> Axis physically blocked or position comparison of motor encoder does not equal external position encoder. **Error correction** Check the parameter Axes->ParameterSets->????->CfgEncoderMonitor->movementThreshold. "????" designates the present parameter set name. - Inform your service agency
230-019A	**Error message** Standstill monitoring err. in %2 **Cause of error** The position error at standstill is greater than the parameter Axes->ParameterSets->????->CfgControllerAuxil->checkPosStandstill defined in the configuration datum. "????" designates the present configuration set name. **Error correction** Inform your service agency.
230-019B	**Error message** %2 does not attain the control window **Cause of error** The configuration datum Axes->ParameterSets->????->CfgControllerTol->posTolerance is defined too low. "????" designates the current parameter set name **Error correction** Increase the value
230-019C	**Error message** Following error in switched-off axis (%2) is too large **Cause of error** Disconnected axis was moved. **Error correction** When an axis is switched off, it must be locked. Or Switch off the parameter Axes->ParameterSets->????->CfgControllerAuxil->driveOffLagMonitor. "????" stands for the current parameter block name.

Error number	Description
230-019D	**Error message** Probe system not ready **Cause of error** - Touch probe is not connected. - Battery in touch probe is dead. - No connection between infrared probe system and receiver unit. **Error correction** - Connect the touch probe. - Exchange the battery. - Clean the receiver unit. To be able to exchange the defective touch probe: 1. In the Manual operating mode, press the "touch probe monitoring" soft key. 2. Delete the error message. 3. Call another tool. Note: The touch probe monitoring is inactive until the next tool call or measuring operation. This means that the NC will not detect a collision with the touch probe!
230-019E	**Error message** Exchange touch probe battery **Cause of error** The battery in the touch probe is dead. **Error correction** Use a fresh battery.
230-019F	**Error message** CC index for %1 too large **Cause of error** In the hardware equipment of this control, there are fewer speed controller processors than were configured for this axis. **Error correction** Check the parameters in the Axes->ParameterSets->????->CfgAxisHardware entity. "????" stands for the current parameter block name.
230-01A0	**Error message** Axis index on CC too large **Cause of error** The axis index on the CC is specified by the parameter selEncoderIn, but the CC has fewer axes than were configured. **Error correction** Distribute the axes into two or more CCs (if available).

Error number	Description
230-01A1	**Error message** Input for position encoder (%1) not found **Cause of error** The input of the position encoder in the indicated axis is configured incorrectly. **Error correction** Check the configuration of the axis: - CfgAxisHardware/posEncoderInput
230-01A2	**Error message** EnDat encoder (%2) reports error **Cause of error** Connected EnDat encoder or encoder cable is defective **Error correction** Check the EnDat encoder or encoder cable
230-01A3	**Error message** Absolute actual position of the axis (%2) not confirmed **Cause of error** Current EnDat position will not be transferred (user input) **Error correction** Check the EnDat encoder or encoder cable, exchange encoder if necessary
230-01A4	**Error message** EnDat encoder (%2) reports different resolution **Cause of error** The resolution reported by the connected EnDat encoder does not match the resolution defined in the configuration data **Error correction** Check the configuration data of the encoder
230-01A5	**Error message** EnDat encoder (%2) reports wrong position **Cause of error** Connected EnDat encoder or encoder cable is defective **Error correction** Check the EnDat encoder or the encoder cable

Error number	Description
230-01A6	**Error message** %2 does not attain the programmed speed **Cause of error** The configuration datum Axes->ParameterSets->????->CfgControllerTol->speedTolerance is defined too low. "????" designates the current name of the configuration set. **Error correction** Increase the value
230-01AB	**Error message** Drive to be moved (%2) is not switched on. **Cause of error** A drive that is supposed to be moved from an NC program or by PLC positioning is not switched on. **Error correction** Check the PLC program
230-01AC	**Error message** Drive to be moved (%2) is not in the position loop **Cause of error** Nominal position values are being generated from an NC program or by a PLC positioning command for a drive that is not in the position loop. **Error correction** Check the PLC program.
230-01AD	**Error message** Deviation in the switch-on position of axis %2 too large **Cause of error** The switch-on position of this axis deviates more than allowed from the position last saved (CfgReferencing->endatDiff). **Error correction** Check the current position. If required, increase the parameter values.
230-01AE	**Error message** Hardware description for axis %2 has changed Position might not be valid **Cause of error** Hardware description parameters of this axis have been changed. Saved positions are invalid. **Error correction** Check the current position.

Error number	Description
230-01AF	**Error message** Error in encoder configuration of axis %1 **Cause of error** Incorrect encoder configuration of the axis. **Error correction** The encoder configuration does not fit the hardware. Refer to the Technical Manual.
230-01B0	**Error message** Error in spindle positioning (%2) **Cause of error** A spindle positioning movement could not be ended properly. **Error correction** The configuration datum Axes/ParameterSets/[key of the axis]/CfgFeedLimits/m19MaxSpeed is defined too low.
230-01B1	**Error message** MC software does not match CC software **Cause of error** Incorrect combination of CC and MC software. **Error correction** Inform your service agency
230-01B2	**Error message** Configuration of digital axes not possible without CC **Cause of error** Without CC only analog axes can be configured. **Error correction** Inform your service agency or change the configuration.
230-01B3	**Error message** S-RAM contents of axis %2 are invalid. **Cause of error** The axis position values saved in S-RAM are invalid. **Error correction** Check the current position
230-01B4	**Error message** Maximum traverse range of EnDat axis was exceeded. **Cause of error** The axis must be readjusted. **Error correction** Redetermine the parameter CfgReferencing->refPosition

Error number	Description
230-01B5	**Error message** The maximum traverse range of the EnDat axis was exceeded while it was switched off **Cause of error** Check the position of the axis. **Error correction** If necessary, reenter the parameter CfgReferencing->refPosition
230-01B6	**Error message** %2 synchronous window not reached **Cause of error** The configuration datum Axes->ParameterSets->????->CfgControllerTol->syncTolerance is defined too low. "????" designates the current name of the configuration set. **Error correction** Increase the value
230-01B7	**Error message** The encoder of an axis in the position control loop must not be switched (%2). **Cause of error** Before a position encoder can be switched, the drive concerned must be switched off by the PLC. **Error correction** Check the NC program, check the PLC program.
230-01B8	**Error message** Two encoders with EnDat interface cannot be used for one axis (%2). **Cause of error** If an EnDat encoder is configured for an axis, the encoder must be entered in parameter block index 0. No more than one EnDat encoder is allowed per axis. **Error correction** - Change the configuration (sequence of parameter blocks). - Change the hardware configuration (encoders).
230-01B9	**Error message** Input X%2 for axis %3 is already assigned to another axis. **Cause of error** The CfgAxes->ParamSet->..->posEncoderInput parameter refers to an input that is already occupied by another axis. **Error correction** - Check the encoder inputs. - If an axis does not have a position encoder, enter the value "none."

Error number	Description
230-01BA	**Error message** Input X%2 for axis %3 is already assigned to another axis. **Cause of error** The CfgAxes->ParamSet->..->speedEncoderInput parameter refers to an input that is already occupied by another axis. **Error correction** - Check the encoder inputs. - If an axis does not have a speed encoder, enter the value "none."
230-01BB	**Error message** Input X%2 for axis %3 is already assigned to another axis. **Cause of error** The CfgAxes->ParamSet->..->pwmSignalOutput parameter refers to an output that is already occupied by another axis. **Error correction** - Check the wiring. - If an axis does not have a PWM output, enter the value "none."
230-01BC	**Error message** Error during control of an SPI module **Cause of error** An error occurred during transfer of rotary encoder data to or from an SPI module (module %2) **Error correction** - Check the wiring - Inform your service agency
230-01BD	**Error message** Spindle (%2) has no position encoder **Cause of error** A selected function (spindle positioning, synchronism, thread, etc.) requires a position encoder of the spindle concerned, but no such encoder has been configured. **Error correction** Check the NC program

Error number	Description
230-01BE	**Error message** AxisMode and AxisHw for axis (%2) do not match **Cause of error** An illegal combination of Axes->PhysicalAxes->CfgAxis->axisMode and Axes->PhysicalAxes->CfgAxis->axisHw has been configured. Permissible combinations include: - AxisMode = NotActive => axisHw = everything allowed - AxisMode = Active => axisHw = InOutCC axisHw = AnalogMC axisHw = AnalogCC axisHw = DisplayMC axisHw = DisplayCC axisHw = ManualMC axisHw = ManualCC - AxisMode = Virtual => axisHw = None **Error correction** Check the configuration.

Error number	Description
230-01BF	**Error message** Encoder type not permitted for axis (%2) **Cause of error** An impermissible combination of Axes->PhysicalAxes->CfgAxis->axisHw and Axes->ParameterSets->CfgAxisHardware->posEncoderType is configured. Permissible combinations are: axisHw = InOutCC CC422 - Motor encoder and all position encoders connected to MC CC424 - Motor encoder and all position encoders connected to CC CC520 - Motor encoder and all position encoders connected to CC axisHw = AnalogMC All position encoders connected to MC axisHw = AnalogCC All position encoders connected to CC axisHw = DisplayMC All position encoders connected to MC axisHw = DisplayCC All position encoders connected to CC axisHw = ManualMC All position encoders connected to MC axisHw = ManualCC All position encoders connected to CC axisHw = ProfiNet Motor encoder and all position encoders connected to ProfiNet **Error correction** Check the configuration
230-01C0	**Error message** Analog output of axis %2 was assigned twice **Cause of error** More than one axis is trying to write to an analog output at one time. **Error correction** Check the PLC program. If more than one axis uses the same analog output, only one at a time can be switched on.
230-01C1	**Error message** Unknown touch probe model designation **Cause of error** An unknown touch probe model designation was selected in the touch probe table. **Error correction** Check the touch probe table.

Error number	Description
230-01C2	**Error message** Spindle (%2) has not yet been homed **Cause of error** A command has been given for a synchronous spindle run. However, not all spindles have been homed. The synchronous spindle run can be switched off only if all involved spindles have been referenced. **Error correction** — Home the spindle - Check the NC or PLC program
230-01C3	**Error message** EMERGENCY STOP defective (%2) **Cause of error** The internal or external EMERGENCY STOP circuit is defective. - Excessively long switching times of the involved relays in the chain between the control-is-ready output signal (MC.RDY, STO.A.G) and the control-is-ready acknowledgment (I3, ES.A, ES.B) during the EMERGENCY STOP or switch-on routine - Control-is-ready acknowledgement (I3, ES.A, ES.B) has a short circuit to +24 V **Error correction** Check the emergency-stop circuit: - Check / replace the applicable relays in the electrical cabinet - Check / restore the contacts / wiring - Inform your service agency
230-01C4	**Error message** Relay ext. dc voltage missing **Cause of error** Error message after power interruption. **Error correction** - Switch on the control voltage separately - Check the wiring in the electrical cabinet - Check the 'Machine control voltage ON' button
230-01C5	**Error message** After SW download, CC no longer responds **Cause of error** After a download of the controller software, an existing CC no longer answers. **Error correction** CC defective. Exchange the hardware.

Error number	Description
230-01C6	**Error message** The option for double-speed control loops has not been enabled **Cause of error** A double-speed control loop was configured, but the option was not enabled. Single-speed controller performance was activated for the control loop. **Error correction** Check the parameter CfgAxisHardware->ctrlPerformance.
230-01C7	**Error message** Communication between MC and CC is faulty **Cause of error** An error occurred in the HSCI communication between the MC computer unit and the CC controller unit. **Error correction** Inform your service agency
230-01C8	**Error message** Timeout during command processing by the CC **Cause of error** A CC was given commands and has not acknowledged them. **Error correction** - Note further messages - Correct the configuration error - If this error occurs without configuration errors: The CC controller unit might be defective. Replace the hardware.
230-01C9	**Error message** Option for master-slave torque control not enabled **Cause of error** You have configured a master-slave torque control, but you have not enabled the required software option. **Error correction** - Check the parameter CfgAxisCoupling - Enable the software option
230-01CA	**Error message** Switch off after NC stop due to error %2 **Cause of error** The machine was switched off after NC stop. Reason: CC error **Error correction** Note the information on remedies while the CC error is displayed.

Error number	Description
230-01CB	**Error message** Output of axis %2 was assigned twice **Cause of error** More than one axis is trying to write to an output at one time. **Error correction** - Check the PLC program If more than one axis uses the same output, only one at a time can be switched on.
230-01CC	**Error message** Input of axis %2 was assigned twice **Cause of error** Two or more axes are trying to read one input at the same time. **Error correction** - Check the PLC program If more than one axis uses the same input, only one at a time can be switched on.
230-01CD	**Error message** Incorrect connector assignment of the axis %2 **Cause of error** On the CC 424 or CC 61xx there is a fixed assignment of speed encoder input to PWM output. The parameters speedEncoderInput and pwmSignalOutput have an illegal connector assignment. Permissible connector assignments: X15 - X51 X16 - X52 X17 - X53 X18 - X54 X19 - X55 X20 - X56 X80 - X57 X81 - X58 X82 - X59 X83 - X60 **Error correction** Check the axis configuration and edit it if required.
230-01CE	**Error message** Machine parameters were changed through TNCOPT **Cause of error** **Error correction**

Error number	Description
230-01CF	**Error message** Switch off after NC stop due to error %2 **Cause of error** The machine was switched off after an NC stop. Reason: PLC error **Error correction** Note the further information on remedies available when the PLC error is displayed.
230-01F5	**Error message** Probe cycle started with stylus already deflected **Cause of error** You tried to start a probing cycle although the stylus is still deflected. **Error correction** Increase the retraction path
230-01F6	**Error message** LookAhead: Time out **Cause of error** Run-time error in LookAhead. **Error correction** Inform your service agency
230-01F7	**Error message** High-speed inputs were incorrectly configured **Cause of error** Only inputs I0 to I31 and I128 to I152 can be used. **Error correction** Check the configuration
230-0226	**Error message** Client with this thread ID is already logged on with CfgServer. **Cause of error** **Error correction** Inform your service agency.
230-0227	**Error message** Configuration server not ready **Cause of error** **Error correction** Inform your service agency.

Error number	Description
230-0228	**Error message** Missing entity (%2) in configuration data **Cause of error** A required parameter is missing in the configuration. **Error correction** Check the configuration.
230-0229	**Error message** Missing entity (%2) in configuration data for axis %3 **Cause of error** A required parameter is missing in the configuration. **Error correction** Check the configuration. If more than one parameter block is assigned to this axis, no other block than block 0 must be complete. However, the extended parameter block key must be entered in configuration date CfgKeySynonym->key and the corresponding basis block key in configuration date CfgKeySynonym->relatedTo.
230-022A	**Error message** Power module %2 not found in table **Cause of error** The specified power module is not listed in the power module table. **Error correction** - Inform your service agency - Check the name of the motor and the power module in the table
230-022B	**Error message** Power module table cannot be read **Cause of error** Cannot read or find the power module table. **Error correction** - The SQL server was given a table file with a syntactically incorrect file name. The file name of the table must begin with a letter, as in M123.D. Change the table's file name. - Check the directory of the power module table - Check the power module table.

Error number	Description
230-022C	**Error message** Motor (%2) not found in motor table **Cause of error** The entered motor is not in the motor table. **Error correction** - Check the motor assignment of the axes - Check the entries in the motor table, especially whether the MODE column has the right value - Inform your service agency
230-022D	**Error message** Motor table (%2) cannot be read **Cause of error** The specified motor table could not be found or read. The file name specified for the motor table is syntactically incorrect. For the control's SQL server to read the file, the file name of the table must contain at least one letter at the beginning. Example: MOTOR123.MOT Observe other displayed messages on the cause of the error. **Error correction** - Correct the file name of the motor table - Check the directory of the motor table - Check the motor table - Check whether the motor table has all required columns - Inform your service agency
230-022E	**Error message** No connection to the SQL server **Cause of error** No connection to the SQL server **Error correction** Inform your service agency.
230-022F	**Error message** Axis (%2) not found in compensation table **Cause of error** Data of the specified axis could not be found in the compensation table. **Error correction** - Check the compensation table. The specified axis must be available as a column in the compensation table. - Inform your service agency.

Error number	Description
230-0230	**Error message** Syntax error in compensation table (%2) **Cause of error** The data in the specified compensation table could not be read. **Error correction** - Check the configuration table (*.cma) - Check the compensation table (*.com) The table can contain up to 1024 compensation points (lines). The AXISPOS column of the first and last line must show the beginning and end of the compensation range with respect to the machine datum. The compensation points between them are calculated internally by the control and do not need to be specified. If you enter optional position values in the AXISPOS column they must have equal spacing. If required, in the BACKLASH column enter compensation values that are measured in negative traverse direction. In the column of the associated axis, enter the values belonging to the compensation points. - Inform your service agency
230-0231	**Error message** Compensation table (%2) cannot be read **Cause of error** Could not find or read the given compensation table. **Error correction** - Check the name and directory of the configuration table saved in the configuration editor through the keyword "TABCMA". - Check the directory of the compensation table saved in the configuration editor through the keyword "oemTable". - The tables assigned to the axes in the configuration table must be in the directory of the compensation tables. - The SQL server was given a table file with a syntactically incorrect file name. The file name of the table must begin with a letter, as in M123.D. Change the table's file name. - Check the configuration table (*.cma) - Check the compensation table (*.com) - Inform your service agency
230-0232	**Error message** Parameter block name (%2) for axis (%3) is already assigned **Cause of error** Two or more axes are defined with reference to the same parameter block. **Error correction** Each axis needs its own parameter block name.

Error number	Description
230-0233	**Error message** Too many parameter blocks for axis %2 **Cause of error** More parameter blocks were requested for an axis than allowed. **Error correction** Make fewer parameter blocks for this axis.
230-0234	**Error message** Shut down the control and restart after you delete an entity **Cause of error** An axis parameter entity was deleted. **Error correction** Restart the control.
230-0235	**Error message** Positioning of axis %2 was stopped due to reconfiguration. **Cause of error** An attempt was made to change a parameter for a moving axis. **Error correction** Axis was stopped.
230-0236	**Error message** Timeout while stopping axis %2 **Cause of error** An attempt was made to change a parameter for a moving axis. **Error correction** Axis was stopped.
230-0237	**Error message** A spindle must be configured as a modulo axis (axis %2) **Cause of error** An axis was defined as spindle that was not configured as a modulo axis. A spindle must always be configured as a modulo axis. **Error correction** Check and, if necessary, correct the parameter CfgAxis->isModulo

Error number	Description
230-0238	**Error message** Fatal configuration error: Cycle machining has been stopped **Cause of error** A fatal error in the configuration has prevented normal operation of the control. **Error correction** Check the configuration
230-0239	**Error message** Name (%2) for axis (%3) is invalid **Cause of error** Two or more axes refer to the same axis key, or an invalid key was installed in System->CfgAxes->axisList. **Error correction** The names must be unambiguous and valid for each axis.
230-023A	**Error message** Invalid configuration for axis %2 **Cause of error** This error can have several causes: 1. In CfgAxis->axisMode, NotAllowed is configured 2. In CfgAxis->axisMode, Active is configured, but this axis has no parameter block 3. In CfgAxis->axisMode, Virtual is configured, but something other than None is in CfgAxis->axisHw **Error correction** Check the combination of parameters
230-023B	**Error message** Fatal error in interpolator: Cyclic machining has been stopped **Cause of error** A fatal error in the interpolator has prevented normal operation of the control. **Error correction** Inform your service agency.
230-023C	**Error message** Supply module %2 not found in table **Cause of error** The specified supply module is not listed in the supply module table. **Error correction** - Inform your service agency. - Check the name of the supply module in the table

Error number	Description
230-023D	**Error message** Supply-module table cannot be read **Cause of error** Cannot read or find the supply module table. The control cannot read or find the supply module table. The file name must begin with a letter. **Error correction** - Check the path of the supply module table under CfgTablePath (SUPPLY or SUPPLY_OEM keys) - Check the directory of the supply module table - Check the supply module table and correct the file name if required.
230-02BC	**Error message** No oriented spindle stop of homed spindle **Cause of error** Oriented spindle stop should be performed with a spindle that has not yet been homed. **Error correction** - Check the NC program - Home the spindle - The parameter Axis->ParamSets->(Spindle)->CfgReferencing->refType must be set to "without Switch and on the fly" so that the spindle automatically homes.
230-02BD	**Error message** Stylus already in contact **Cause of error** The stylus is already deflected at the start of a probing movement. **Error correction** - Get the touch probe clear and repeat the probe. - If the error frequently recurs, inspect the probe for damage. - If necessary, contact your service agency.
230-02BF	**Error message** Handwheel? **Cause of error** - The electronic handwheel is not connected. - An incorrect handwheel is configured in machine parameter System/CfgHandwheel/type. - The transmission line is defective was incorrectly chosen. **Error correction** - Connect the handwheel via cable adapter. - Check the machine parameter System/CfgHandwheel/type. - Inspect the transmission line for damage

Error number	Description
230-02C0	**Error message** Velocity programmed for axis %2 too low **Cause of error** The velocity programmed for this axis for PLC positioning is too slow. **Error correction** Program a faster velocity or check the configuration datum Axes->ParameterSets->????->CfgFeedLimits->minFeed. "????" designates the current name of the configuration set.
230-02C1	**Error message** Rotational speed programmed for spindle ("%2") too low **Cause of error** The shaft speed programmed for this axis for PLC positioning is too slow. **Error correction** Program a faster shaft speed or check the configuration datum Axes->ParameterSets->????->CfgFeedLimits->minFeed. "????" designates the current name of the configuration set.
230-02C2	**Error message** Return to contour in thread cycle not allowed. **Cause of error** In a thread cycle, you tried to return in the tilted working plane. **Error correction** In thread cycles, always return only in the tool axis direction.
230-02EE	**Error message** The option for HSC filter has not been enabled **Cause of error** An HSC filter was configured, but the option was not enabled. Now the triangle filter was activated for this axis. **Error correction** Configure another file type

Error number	Description
230-02EF	**Error message** Software option 151 Load Monitoring not enabled **Cause of error** A load monitor (G995, G996) was programmed in the NC program, but the required software option 151 Load Monitoring is not enabled. **Error correction** - Check the NC program and adapt it if necessary - If required, enable the software option 151 Load Monitoring - Contact the machine tool builder - Inform your service agency
230-02F0	**Error message** Option for digital control loops not enabled **Cause of error** A digital control loop was configured without enabling the necessary option in the control's SIK. **Error correction** - Check the configuration and correct if necessary - Enable the option - Inform your machine tool builder
230-0327	**Error message** Plug & Play (%2) Motor detected: %3 **Cause of error** The "Plug & Play" function for automatic recognition of drive components through the electronic ID label is active for this axis. A motor was recognized that differs from the current configuration. **Error correction** - Confirm the "Plug & Play" dialog, if the detected motor belongs to this combination of axis and parameter set - Deactivate "Plug & Play", if the motor was not correctly recognized - Contact your machine tool builder

Error number	Description
230-0328	**Error message** Plug & Play (%2) Inverter detected: %3 **Cause of error** The "Plug & Play" function for automatic recognition of drive components through the electronic ID label is active for this axis. An inverter was recognized that differs from the current configuration. **Error correction** - Confirm the "Plug & Play" dialog, if the detected inverter belongs to this combination of axis and parameter set - Deactivate "Plug & Play", if the inverter was not correctly recognized - Contact your machine tool builder
230-0329	**Error message** Plug & Play (%2) Supply Module detected: %3 **Cause of error** The "Plug & Play" function for automatic recognition of drive components through the electronic ID label is active. An supply module was recognized that differs from the current configuration. **Error correction** - Confirm the "Plug & Play" dialog prompt if the supply module was correctly recognized - Deactivate "Plug & Play", if the supply module was not correctly recognized and check the configuration - Contact your machine tool builder
230-032A	**Error message** Axis %1 (%2): Motor %3 detected and registered **Cause of error** The named motor was detected through plug-and-play and entered in the axis configuration under CfgServoMotor->motName. **Error correction**
230-032B	**Error message** Axis %1 (%2): Inverter %3 detected and registered **Cause of error** The named inverter was detected through plug-and-play and entered in the axis configuration under CfgPowerStage->ampName. **Error correction**

Error number	Description
230-032C	**Error message** Axis %1 (%2): Supply module %3 detected and registered **Cause of error** The named supply module was detected through plug-and-play and entered in the axis configuration under CfgSupply-Module->name. **Error correction**
230-032D	**Error message** Axis %1 (%2): Plug & play has been deactivated for the motor **Cause of error** Plug-and-play was deactivated in the named axis for the motor. **Error correction** - Check the configuration and the motor being used - Plug-and-play can be reactivated through CfgServoMo-tor->plugAndPlay
230-032E	**Error message** Axis %1 (%2): Plug & play has been deactivated for the invert-er **Cause of error** Plug-and-play was deactivated in the named axis for the inverter. **Error correction** - Check the configuration and the inverter being used - Plug-and-play can be reactivated through CfgPowerS-tage->plugAndPlay
230-032F	**Error message** Axis %1 (%2): Plug & play has been deactivated for supply module **Cause of error** Plug-and-play was deactivated in the named axis for the supply module. **Error correction** - Check the configuration and the supply module being used - Plug-and-play can be reactivated through CfgSupplyMod-ule->plugAndPlay
230-041A	**Error message** Position error (axis %2) too large **Cause of error** The difference between position measurements by the position encoder and speed encoder is too large. **Error correction** Check the position and speed encoders.

Error number	Description
230-041B	**Error message** Axis %2 cannot be switched on **Cause of error** This axis is to be switched on by the PLC, although it was switched off by DriveOffGroup. **Error correction** Check the PLC program.
230-041C	**Error message** Error bit in safety status (S status) of the HSCI transmission **Cause of error** An error was signaled over the HSCI S status. **Error correction** - Inform your service agency. - You can find more diagnostic information in the diagnostics menu.
230-041D	**Error message** TRC: Wrong control; axis %1 **Cause of error** The compensation file was generated for another control than the one now in use. Copying the compensation file from another control is not allowed. **Error correction** - Recalculate the compensation parameter with TNCopt under Optimization/Torque Ripple Compensation. - Deactivation of the compensation: Entry in configuration datum Go to "Axes/ParameterSets/[Keyname of parameter block]/CfgControllerComp/" and delete "compTorqueRipple." - Inform your service agency
230-041E	**Error message** TRC: Compensation file (%1) illegible **Cause of error** Could not find or read the given compensation file. **Error correction** - Check the directory of the compensation file saved in the configuration editor through the keyword "oemTable". - Check the compensation file. - Deactivation of the compensation: Entry in configuration datum Go to "Axes/ParameterSets/[Keyname of parameter block]/CfgControllerComp/" and delete "compTorqueRipple." - Inform your service agency

Error number	Description
230-041F	**Error message** Error in parameter posEncoderIncr or posEncoderDist (axis %2) **Cause of error** The two machine parameters posEncoderIncr and posEncoderDist in the configuration object CfgAxisHardware are configured incorrectly. Even if you operate the axes solely with motor encoders (without linear encoders), the two parameters must contain realistic values. **Error correction** - Enter in the machine parameters CfgAxisHardware->posEncoderIncr and CfgAxisHardware->posEncoderDist realistic values for the position or motor encoder.
230-0420	**Error message** No field angle for drive %1 **Cause of error** The field angle of a motor with an unaligned encoder has not yet been ascertained. Absolute encoder with EnDat interface: - The encoder serial number saved does not fit the encoder Incremental encoders: - The SIK serial number saved does not fit the SIK of the control **Error correction** - If required, find the field angle in the "current controller adjustment" mode (press the "FIELD ORIENT." soft key) - Check the “type of encoder” entry in the motor table and correct it if necessary - Check the machine parameter motEncType and correct it if required - Check the machine parameters motPhiRef and motEncSerialNumber. If you want to force a new field angle acquisition, enter the value 0. - Inform your service agency
230-0421	**Error message** MCU/CCU watchdog mismatch **Cause of error** MCU and CCU have different watchdog values. **Error correction** Inform your service agency.

Error number	Description
230-0422	**Error message** Axis motion not allowed while the filter is being changed **Cause of error** All axes must remain stationary while the filter is being edited. **Error correction** Inform your service agency.
230-0423	**Error message** Error in configuration of axis %2 **Cause of error** Connections were configured that are not available on this CC. **Error correction** Check the configuration of the axis: - CfgAxisHardware->posEncoderInput - CfgAxisHardware->speedEncoderInput - CfgAxisHardware->inverterInterface For analog axes via CMA-H expansion module: - Check whether the expansion module is functioning correctly
230-0424	**Error message** DCM: %1 **Cause of error** Dynamic collision monitoring (DCM) stopped all axis movements in order to avoid a collision. **Error correction**
230-0425	**Error message** Unbalance monitoring: measurement not possible **Cause of error** The measurement of the unbalance failed. The spindle could not be accelerated correctly. The programmed nominal speed was not attained. **Error correction** - Inspect the spindle for damage. - Inform your service agency.
230-0426	**Error message** Unbalance monitoring (user): unbalance too large Unbalance monitoring (system): unbalance too large **Cause of error** The value found by the unbalance monitor was too large. **Error correction** - Balance the signal

Error number	Description
230-0427	**Error message** Unbalance monitoring (user): sum of unbalance too large Unbalance monitoring (system): sum of unbalance too large **Cause of error** The unbalance sum calculate by the unbalance monitor was too large. **Error correction** - Balance the signal
230-042A	**Error message** Unbalance monitoring: configuration missing **Cause of error** The machine parameters for configuring the unbalance monitoring are not available. **Error correction** - Check the config object CfgUnbalance and correct it if required.
230-042B	**Error message** Unbalance monitoring: invalid spindle defined **Cause of error** No value spindle index was given in the machine parameters for unbalance monitoring. **Error correction** - Check the machine parameters in the config object CfgUnbalance and correct it if required.
230-042C	**Error message** Unbalance monitoring: invalid meas. axis defined **Cause of error** An invalid measuring axis was given in the machine parameters for unbalance monitoring. **Error correction** - Check the machine parameter axisOfMeasure in the config object CfgUnbalance and correct it if required.
230-042D	**Error message** Unbalance monitoring: system monitor is not active **Cause of error** The system monitor of the unbalance monitoring is not active. **Error correction** - Activate the system monitor through the turning cycle - Check the machine parameters maxUnbalanceOem and limitUnbalanceOem and correct it if required.

Error number	Description
230-042E	**Error message** Unbalance monitoring: spindle is not configured **Cause of error** The machine parameters for configuring the spindle are missing for the unbalance monitor. **Error correction** - Check the configuration of the spindle in the config object CfgUnbalance and correct it if required.
230-042F	**Error message** Unbalance monitoring: spindle parameter or index is faulty **Cause of error** The spindle configuration or configuration of the unbalance monitor is faulty. An incorrect spindle index was given or the spindle configuration in the config object CfgUnbalance is incorrect. **Error correction** - Check the spindle index in the CfgAxes config object and correct it if required. - Check the machine parameters in the config object CfgUnbalance and correct it if required.
230-0430	**Error message** Unbalance monitoring: limit speed has been reached **Cause of error** The maximum permissible shaft speed calculated by the control for measuring the unbalance was attained. **Error correction** - Reduce the speed of the spindle and restart the measurement.
230-0431	**Error message** Unbalance trace: internal error **Cause of error** Internal error in unbalance trace: incorrect status in the IPO in unbalance monitoring **Error correction** - Inform your service agency.
230-0432	**Error message** Unbalance trace: trigger timeout **Cause of error** In the unbalance trace, the trigger conditions defined in the machine configuration were not fulfilled. **Error correction** - Check the machine parameters triggerMin and triggerMax, and correct it if required.

Error number	Description
230-0433	**Error message** Unbalance trace: internal error **Cause of error** An incorrect trace channel is active for the unbalance trace. **Error correction** - Inform your service agency.
230-0434	**Error message** Unbalance trace: spindle index refers to an incorrect axis **Cause of error** An incorrect spindle index was given for the unbalance trace. The index refers to an incorrect address. **Error correction** - Check the machine parameters in the config object CfgUn-balance and correct it if required.
230-0435	**Error message** Unbalance trace: machine parameters not defined **Cause of error** The machine parameter for configuring the unbalance traces are not available. **Error correction** - Enter machine parameters in the config object CfgUnbal-ance.
230-0436	**Error message** Unbalance trace: internal error **Cause of error** The OEM cycle for the unbalance trace is faulty. The shaft speed or the number of revolutions to be measured was not defined. The parameter count or speed must not have the value 0. **Error correction** - Correct the OEM cycle for the unbalance trace. The values for count or speed must not be 0.
230-0437	**Error message** Unbalance trace: incorrect spindle index **Cause of error** An invalid spindle index was given for the unbalance trace. **Error correction** - Check the machine parameter for the spindle and correct if required.

Error number	Description
230-0438	**Error message** Unbalance trace: axis for measurement not defined **Cause of error** No measuring axis was defined for the unbalance trace. **Error correction** - Check the machine parameter axisOfMeasure and correct if required.
230-0439	**Error message** Unbalance trace: spindle is not configured **Cause of error** The spindle defined for the unbalance trace was not found in the machine configuration. **Error correction** - Check the machine parameter for the spindle and correct if required.
230-043A	**Error message** Unbalance trace: specified trigger axis does not exist **Cause of error** The trigger axis defined for the unbalance trace does not exist. **Error correction** - Check the machine parameter for the spindle and correct if required.
230-043B	**Error message** Unbalance trace: trigger conditions are not fulfilled **Cause of error** The trigger conditions set for the unbalance trace are no longer fulfilled. **Error correction** - Check the machine parameters triggerMin and triggerMax, and correct it if required.
230-043C	**Error message** Unbalance monitoring: permissible unbalance not defined **Cause of error** A user parameter required for the unbalance monitoring (the maximum permissible unbalance) was not defined. **Error correction** - Enter the user parameter maxUnbalanceUsr.

Error number	Description
230-043D	**Error message** Unbalance monitoring: permissible unbalance sum not defined **Cause of error** A user parameter required for the unbalance monitoring (the maximum permissible unbalance sum) was not defined. **Error correction** - Enter the user parameter limitUnbalanceUsr.
230-043E	**Error message** Unbalance monitoring: permissible unbalance not defined **Cause of error** A machine parameter required for the unbalance monitoring, the maximum permissible unbalance (system-wide), was not defined. **Error correction** - Enter the machine parameter maxUnbalanceOem.
230-043F	**Error message** Unbalance monitoring: permissible unbalance sum not defined **Cause of error** A machine parameter required for the unbalance monitoring, the maximum permissible unbalance sum (system-wide), was not defined. **Error correction** - Enter the machine parameter limitUnbalanceOem.
230-0441	**Error message** Internal error in transmission of oscilloscope measured data **Cause of error** The display of measured value in the oscilloscope is incomplete because of an error in the internal data transmission between the interpolator real-time buffer and the oscilloscope. **Error correction** - Repeat the measurement - Inform your service agency if the error continues to occur

Error number	Description
230-0442	**Error message** Progmmd. limit switch/protection zone incorrect for axis %2 **Cause of error** The following constraints apply to limit switches and protection zones for modulo axes: - The lower limit must be between –360° and +360°. - The upper limit must be between 0° and +360°. - The lower limit must be less than the upper limit. - The difference between the upper and lower limit must be less than 360°. - If the machine parameter "moveAfterRef" is configured, the axis is not moved further. A warning is output. - Both protection zones = 0 means: Monitoring is inactive **Error correction** Adjust the value for the protection zone.
230-0443	**Error message** Handwheel superimposition not allowed (M118) **Cause of error** You tried to activate the M118 function during active collision monitoring. The handwheel superimpositioning function with M118 is not allowed in combination with collision monitoring. **Error correction** Remove M118 from the NC program or deactivate the collision monitoring.
230-0444	**Error message** Collision monitoring not possible in turning mode **Cause of error** The collision monitoring cannot monitor turning tools and collision objects that rotate with the lathe spindle. **Error correction** - If necessary, remove the collision objects (CMOs) from the turning kinematics (notify the machine manufacturer). - Do not insert any turning tools during milling.
230-0445	**Error message** Velocity error (axis %2) too great **Cause of error** The velocity deviation between position the position encoder and the speed encoder is too large. **Error correction** Check the coupling between the position encoder and speed encoder

Error number	Description
230-0446	**Error message** Internal error while processing the oscilloscope measuring data **Cause of error** Internal software error: The synchronization of the oscilloscope channels is faulty. **Error correction** - Repeat the measurement - Inform your service agency if the error continues to occur
230-0447	**Error message** Internal error while processing the oscilloscope measuring data **Cause of error** Internal software error: The synchronization of the oscilloscope channels in combination with the trigger condition is faulty. **Error correction** - Repeat the measurement - Inform your service agency if the error continues to occur
230-0448	**Error message** Internal error while processing the oscilloscope measuring data **Cause of error** Internal software error: The sequence of the data to be transmitted to the oscilloscope is faulty. **Error correction** - Repeat the measurement - Inform your service agency if the error continues to occur
230-0449	**Error message** Handwheel: wrong handwheel connected Handwheel %3: Wrong handwheel connected **Cause of error** - The electronic handwheel is not connected. - An incorrect handwheel is configured in machine parameter System/CfgHandwheel/type. **Error correction** - Connect the handwheel via cable adapter. - Check the machine parameter System/CfgHandwheel/type.

Error number	Description
230-044A	**Error message** Handwheel: Contaminated or damaged Handwheel %3: Contaminated or damaged **Cause of error** The handwheel reports a problem in signal transmission: - Rotary encoder in the handwheel is contaminated - Handwheel is defective **Error correction** - Check the emergency stop and permissive buttons for proper function - Exchange handwheel if necessary
230-044B	**Error message** Handwheel: Transmission disturbance Handwheel %3: Transmission disturbance **Cause of error** - An incorrect handwheel was configured in machine parameter System/CfgHandwheel/type. - Data transfer between the handwheel and control was disturbed. **Error correction** - Check the machine parameter System/CfgHandwheel/type. - For wireless handwheel: Reduce distance to receiver - Switch off possible sources of interference - Check the connecting cable
230-044C	**Error message** Handwheel: Transmission error Handwheel %3: Transmission error **Cause of error** The transmission line is defective or faulty. **Error correction** Inspect the data transfer line for damage.
230-044D	**Error message** Handwheel: Wrong parameter Handwheel %3: Wrong parameter **Cause of error** The initialization values for the connected handwheel are invalid. **Error correction** Check the configuration datum System/CfgHandwheel/initValues.

Error number	Description
230-044E	**Error message** Handwheel: Timeout Handwheel %3: Timeout **Cause of error** A time limit was exceeded during communication with the handwheel. **Error correction** - Check the access point of the handwheel - Check the radio settings
230-044F	**Error message** Handwheel: No connection possible Handwheel %3: No connection possible **Cause of error** Cannot connect with the handwheel. The handwheel might not be in the access point (handwheel holder). **Error correction** Place the handwheel in the access point (handwheel holder).
230-0450	**Error message** Axis %2: Enter CfgReferencing/doubleRefOffset: %3 **Cause of error** The reference run was performed twice. **Error correction** Enter the indicated value in the machine configuration (parameter: CfgReferencing/dblRefOffset).
230-0451	**Error message** PLC movement for axis %2 not allowed **Cause of error** The presently active NC program blocks manual axis movements by the manual direction keys or movements by the PLC program. **Error correction** Edit the PLC Program
230-0452	**Error message** DCM: No manual movement allowed during a program run **Cause of error** You tried to make a movement with axis direction keys, a handwheel or PLC command while a program run was active **Error correction** Wait until the program run has ended or change to Single Block mode

Error number	Description
230-0453	**Error message** DCM: Program start or continuation not possible **Cause of error** You tried to start a program during an axis movement, e.g. by the axis direction key, handwheel or PLC command. **Error correction** Wait until the motion by axis direction key, handwheel or PLC command has finished
230-0454	**Error message** DCM: No TCPM allowed during active DCM **Cause of error** You tried to move in Manual mode with TCPM during active DCM **Error correction** Deactivate TCPM Deactivate DCM and move without monitoring
230-0455	**Error message** DCM: Motion of the floating tap holder is not monitored **Cause of error** When DCM is active, you use the "tapping with floating tap holder" cycle. Please note that movements of the floating tap holder are not monitored by DCM. Collision monitoring observes the floating tap holder at its resting position. **Error correction**
230-0456	**Error message** DCM not possible in following error mode or semifeedforward mode **Cause of error** You tried to use DCM with a non-feedforward axes. **Error correction** Change the configuration
230-0457	**Error message** Reference mark not found **Cause of error** The reference mark was not found after covering the distance necessary for distance-coded referencing. **Error correction** - Inform your machine tool builder - Check the mounted encoder - Check the machine configuration (parameter "posEncoderRefDist")

Error number	Description
230-0458	**Error message** DCM: Check the skipReferencing parameter **Cause of error** You have set the machine parameter "System/CfgMachineSimul/skipReferencing" to the value TRUE. The Dynamic Collision Monitoring (DCM) is not possible with this setting. **Error correction** Set the parameter "skipReferencing" to the value FALSE or activate the programming-station mode (simMode = CcAndExt)
230-0459	**Error message** S-RAM contents of axis %2 are inconsistent **Cause of error** The EnDat axis position values saved in S-RAM are invalid. The values saved in a file will be used. **Error correction** Check the current position of the axis
230-045B	**Error message** DCM: Activation during a movement **Cause of error** You activated the dynamic collision monitoring (DCM) during a program run or an axis movement. **Error correction** Activate DCM when the machine is at standstill.
230-045C	**Error message** S-RAM contents of axis %2 overwritten with values from file **Cause of error** The EnDat axis position values saved in S-RAM were overwritten by the values saved in a file. **Error correction** Check the current position of the axis
230-045D	**Error message** Axis %2: CfgReferencing/doubleRefOffset: %3 was entered **Cause of error** The reference run was performed twice. **Error correction** The given value was entered in the configuration.

Error number	Description
230-045E	**Error message** Oscilloscope active during start of reference machining **Cause of error** The oscilloscope is active and a reference operation was started at the same time. The oscilloscope cannot be used during the reference operation. **Error correction** Close the oscilloscope and repeat the reference operation
230-045F	**Error message** Programmed movement not allowed **Cause of error** No programmed movement of axes is allowed in the "Retract" operating mode. **Error correction** Do not start an NC program as long as the "Retract" operating mode is active.
230-0460	**Error message** Error in configuration of axis %2 **Cause of error** The axis has been configured incorrectly. **Error correction** Check the axis configuration at the following places and correct if required: - CfgSupplyModule->name - CfgPowerStage->ampPowerSupplyType - CfgPowerStage->ampBusVoltage - CfgServoMotor->motSupply
230-0461	**Error message** Configuration of kinematic compensation is faulty **Cause of error** The configuration of the temperature compensation or the kinematic compensation is faulty. The controls traverses without compensation as long as the error is not corrected. The more exact cause is described in the additional text. **Error correction** - Acknowledge the error in order to continue without compensation - Correct the configuration

Error number	Description
230-0462	**Error message** Temperature compensation is faulty **Cause of error** An error occurred in the calculation of the temperature compensation. The temperature compensation might no longer be working. **Error correction** Check the configuration of the temperature compensation.
230-0463	**Error message** Kinematic compensation is faulty **Cause of error** An error occurred in the calculation of the kinematic compensation. The kinematic compensations might no longer be working. **Error correction** Check the kinematic compensations and and correct them if required.
230-0464	**Error message** EnDat multiturn counter of the %2 axis was corrected **Cause of error** The value for the EnDat multiturn counter saved in the configuration is not plausible. The value was automatically corrected by the control. **Error correction** Check the current position of the axis
230-0465	**Error message** EnDat multiturn counter of the %2 axis was changed **Cause of error** The value for the EnDat multiturn counter saved in the configuration was exceeded. The change does not become effective until the control is restarted. **Error correction** Restart the control.
230-0467	**Error message** The KinematicsComp option has not been enabled **Cause of error** An kinematic compensation was configured, but the KinematicsComp option was not enabled. The kinematic compensation is not in effect. **Error correction** - Correct the configuration or enable the software option.

Error number	Description
230-0468	**Error message** Max. compensation value %2 reached in CfgKinSimpleTrans %1 **Cause of error** The value for the kinematic temperature compensation at the transformation has exceeded the maximum permissible value. The compensation is set to the maximum value. The warning is deleted as soon as the maximum value is no longer exceeded. **Error correction** - Check the calculation of the compensation, and correct it if required - Check the incoming PLC variables
230-0469	**Error message** Max. compensation value %2 reached in axis %1 component %3 **Cause of error** The value for the kinematic compensation at the axis has exceeded the maximum permissible value. The compensation is set to the maximum value. The warning is deleted as soon as the maximum value is no longer exceeded. **Error correction** - Contact your machine tool builder - Check the value and correct it if required - Check the incoming PLC variables and tables and correct them if required
230-046A	**Error message** More than %2 fast PLC inputs defined **Cause of error** In the IOC file, more fast PLC inputs are defined than allowed. **Error correction** - Check the IO configuration. - Inform your service agency

Error number	Description
230-046B	**Error message** Fast PLC input on impermissible bus system **Cause of error** - In the IOC file, a fast PLC input was defined on an illegal bus system. - Fast PLC inputs can be defined only on an HSCI-PL or the internal PL. The affected PLC input is shown in the additional information. **Error correction** - Check the IO configuration - Inform your service agency
230-046C	**Error message** Fast PLC inputs with more than one definition **Cause of error** Fast PLC inputs are defined both in the IOC file as well as in the configuration data (machine parameters). Note that the entry in the configuration data has priority. **Error correction** - Check the IO configuration - Delete the CfgPlcFastInput configuration parameter (parameter number 103700) from the configuration data, if required - Inform your service agency
230-046D	**Error message** Input for axis group enabling has incorrect parameters **Cause of error** The PLC input for the axis-group release was not in the IO configuration (IOC file) or it was faulty. **Error correction** - Check the IO configuration. - Inform your service agency
230-046E	**Error message** Multiple inputs defined for axis group enabling %2 **Cause of error** - Per axis group, only one PLC input for the axis-group release was defined. - In the IOC file, multiple PLC inputs for the axis-group release were defined for one axis group. **Error correction** - Check the IO configuration. - Inform your service agency

Error number	Description
230-046F	**Error message** Input for axis group enabling %s on impermissible bus system **Cause of error** The input for the axis-group release was defined on an illegal bus system in the IO configuration (IOC file). The input can be defined only on an HSCI-PL or an internal PL. **Error correction** - Check the IO configuration. - Inform your service agency
230-0470	**Error message** Fast input for spindle has incorrect parameters **Cause of error** The fast input for the spindle was not in the IO configuration (IOC file) or it has incorrect parameters. **Error correction** - Check the IO configuration. - Inform your service agency
230-0471	**Error message** Several fast inputs defined for spindle %2 **Cause of error** More than one fast input was defined for a spindle in the IO configuration (IOC file). Only one input is allowed. **Error correction** - Check the IO configuration. - Inform your service agency
230-0472	**Error message** Fast input for spindle %1 on impermissible bus system **Cause of error** - The fast input for the spindle was defined on an illegal bus system. - The input can be defined only on an HSCI-PL or an internal PL. **Error correction** - Check the IO configuration. - Inform your service agency

Error number	Description
230-0473	**Error message** Fast input cannot be activated **Cause of error** A fast input cannot be activated because the IO configuration does not match the actual hardware configuration. The affected input is shown in the additional information. - The control is operated in the simulation mode. - The IOC file does not match the hardware configuration. - The options in the configuration are incorrectly set. **Error correction** - Check the hardware configuration - Check the IO configuration - Check the options - Inform your service agency
230-0474	**Error message** Inputs for axis group enabling have multiple definitions **Cause of error** Inputs for axis-group release are defined both in the IOC file as well as in the configuration data (machine parameters). Note that the entry in the configuration data has priority. **Error correction** - Check the IO configuration - Delete the configuration parameter driveOffGroupInput (parameter number 100106) if necessary - Inform your service agency
230-0475	**Error message** Fast input for spindle %2 already defined **Cause of error** Fast inputs are defined for the spindle both in the IOC file as well as in the configuration data (machine parameters). Note that the entry in the configuration data has priority. **Error correction** - Check the IO configuration. - If required, delete the configuration datum CfgSpindle->fastInput (parameter number 401502)
230-0479	**Error message** TRC: The identification was adapted; axis %1; file %2 **Cause of error** The compensation file for the TRC was adapted. The values were transferred to the CC controller unit and activated. **Error correction** Note further messages.

Error number	Description
230-047A	**Error message** No fast input for spindle %2 (%3) **Cause of error** No fast input was designed for homing or stopping. **Error correction** - Check the IO configuration (IOC file)
230-047B	**Error message** Programmed position error too large. Limit to %1 mm. **Cause of error** - The configured position error is too large for KinematicsComp. - The configured value is limited. **Error correction** - Correct the corresponding value(s) in the kinematic configuration (machine parameters locErrX, locErrY, locErrZ, locErrA, locErrB, locErrC). - Inform your service agency
230-047E	**Error message** Axis %2: Double reference name is active **Cause of error** A scanning cycle was started with active double reference run although the position encoder's reference mark had not yet been scanned. **Error correction** Traverse the reference mark - Check the entry for double reference run in machine parameter MP_doubleRef - Inform your service agency
230-047F	**Error message** Current actual position of the axis (%2) cannot be used **Cause of error** - The machine tool builder's configuration does not allow transfer of the current EnDat position **Error correction** - Inform your service agency

Error number	Description
230-0480	**Error message** Max. compensation value %2 reached in axis %1 **Cause of error** The value for the kinematic compensation at the axis has exceeded the maximum permissible value. The compensation is set to the maximum value. The warning is deleted as soon as the value falls 0.1 mm below the maximum value. Both the kinematic temperature compensation and the compensation via KinematicsComp (software option) go into the kinematic compensation. **Error correction** - Check calculation of compensation values, and correct if required - Check and the PLC variables and tables going into the compensation and correct them if necessary - Inform your service agency
230-0488	**Error message** No assignment between handwheels and connections **Cause of error** - Two or more handwheels were configured (CfgHandwheel) but no connections assigned - The automatic assignment to a connection is possible only with a single handwheel **Error correction** - Reduce the number of configured handwheels to one handwheel - Assign individual handwheels to one connection each (CfgHandwheelList) - Inform your service agency
230-0489	**Error message** Only one HR 550FS wireless handwheel possible **Cause of error** - More than one active HR 550FS wireless handwheel is configured **Error correction** - Check and adjust the range of action. Only one wireless handwheel can be active: Deactivate wireless handwheels (CfgHandwheel->type) or remove the assignment (CfgHand-wheelList) - Inform your service agency

Error number	Description
230-048C	**Error message** Error in acknowledging the SampleRate of the CC **Cause of error** The acknowledged SampleRate of the CC controller unit does not agree with the SampleRate of the set parameters **Error correction** - Check parameter MP_ampPwmFrq for the SampleRate - Inform your service agency
230-048D	**Error message** Handwheel superimpositioning is ignored **Cause of error** The handwheel superimpositioning cannot be changed during M140 and measuring movements. **Error correction** The handwheel superimpositioning can be changed again when no M140 or measuring movement is active.
230-048E	**Error message** Internal software error during Program Run, Single Block **Cause of error** An internal error occurred that can disturb the block display in Single Block mode. **Error correction** - Delete the error and continue as normal. In very rare cases in Single Block mode the block display might not agree with the operation. The movements are nevertheless made individually. - If the error should reoccur, please generate a service file and inform your service agency.
230-048F	**Error message** Handwheel %3: Communication error **Cause of error** - Internal communication error with the handwheel **Error correction** - Save service files - Inform your service agency
230-0490	**Error message** PLC movement for axis %2 was canceled **Cause of error** - An axis movement through the manual direction keys or the PLC program was stopped by a reconfiguration or a system cycle. **Error correction** - Restart the axis movement, if required.

Error number	Description
230-0491	**Error message** Retraction from thread completed **Cause of error** The retraction from the thread has been concluded. **Error correction** - NC program is to be continued: acknowledge message and continue NC program with NC start. - NC program is not to be continued: acknowledge message and terminate NC program with INTERNAL STOP.
230-0492	**Error message** NC software does not match UVR firmware **Cause of error** Faulty combination of NC software and UVR firmware. **Error correction** Inform your service agency
230-0493	**Error message** MCU/UVR watchdog mismatch **Cause of error** The watchdogs on the MC main computer and UVR supply unit have different values. **Error correction** Generate the service files and inform your service agency.
230-0494	**Error message** Message from UVR %2 **Cause of error** The UVR supply unit reports an error **Error correction** Inform your service agency
230-0495	**Error message** Incorrect axis index in axis %1 **Cause of error** - Machine parameter CfgAxisHardware/MP_ccAxisIndex contains an invalid value. **Error correction** - Check and, if necessary, change the configuration in MP_c-cAxisIndex: Valid values are 0 to N-1, where N is the number of control loops on the controller unit concerned. Example: For a CC xx06, the valid values are 0 to 5. - Inform your service agency

Error number	Description
230-0496	**Error message** Input for speed encoder (%1) not valid **Cause of error** Faulty configuration of the speed encoder input for the axis **Error correction** Check the configuration of the axis: - CfgAxisHardware/MP_speedEncoderInput
230-0497	**Error message** Incorrect inverter or motor connection (%1, inv. %2, motor %3) **Cause of error** - The configured connection for the inverter to the controller unit is not available (CfgAxisHardware/MP_inverterInterface) - or no inverter is connected to the configured socket (CfgAxisHardware/MP_inverterInterface) - or the configured connection for the motor on the inverter is not available (CfgAxisHardware/MP_motorConnector). **Error correction** Check the following machine parameters: - CfgAxisHardware/MP_inverterInterface - CfgAxisHardware/MP_motorConnector
230-0498	**Error message** Faulty synchronization during a movement **Cause of error** A system error led to an incorrect synchronization **Error correction** Inform your service agency
230-0499	**Error message** CfgLaAxis/MP_axManualJerk (%2) missing for interpol. PLC movement **Cause of error** - Machine parameter CfgLaAxis/MP_axManualJerk must be configured for interpolating PLC movements. **Error correction** - For the PLC axes to be moved in interpolation, check the configuration in the machine parameter CfgLaAxis/MP_axManualJerk. - Inform your service agency

Error number	Description
230-049A	**Error message** Spindle speed? **Cause of error** The NC program cannot be simulated because the spindle speed for the feed per revolution is missing. A simulation is possible only with the simulation speed FMAX. **Error correction** - Check the NC program and change it if necessary - Change the simulation speed to FMAX.
230-049C	**Error message** Timeout during command processing by the UVR %2 **Cause of error** A UVR was given a command and has not acknowledged it **Error correction** Possible causes: - HSCI connection interrupted (see other error messages) - UVR defective
230-049D	**Error message** A parameter change requires switching off the drive **Cause of error** Before changing the directional bit in the parameters, the drive must be switched off. **Error correction**
230-049E	**Error message** Real time coupling function (RTC) together with handwheel active **Cause of error** - Real time coupling function (RTC) is to be closed while the handwheel is active, or - the handwheel is to be activated while the real time coupling function is active. **Error correction** - Check the NC program and correct it if necessary - Activate the handwheel at a later time - Inform your service agency
230-04A0	**Error message** Test of brake %1 for axis %2 was unsuccessful **Cause of error** - Note further messages. **Error correction** - Inform your service agency

Error number	Description
230-04A2	**Error message** Touch probe %1 is not supported by the transmitter/receiver unit **Cause of error** The transmitter/receiver unit does not support the touch probe. **Error correction** Select another touch probe Inform your service agency
230-04A3	**Error message** Collision of the touch probe **Cause of error** The touch probe collision protection has responded. **Error correction** Manually retract the touch probe.
230-04A4	**Error message** Configuration of handwheel on axis %2 is faulty **Cause of error** Connections were configured that are not available on this CC. **Error correction** Check the configuration of the axis: - CfgAxisHandwheel->hsciCcIndex - CfgAxisHandwheel->input
230-04A5	**Error message** Handwheel %3: Handwheel is not supported **Cause of error** - The electronic handwheel is not supported with this NC software version **Error correction** - Check the software version - Inform your service agency - If required, use an older-model handwheel - If required, install the service pack of the NC software that supports this handwheel model

Error number	Description
230-04A6	**Error message** Multiple handwheels on the same radio channel **Cause of error** There might be several radio handwheels operating in the environment with the same radio channel. **Error correction** - Check the channels selected for the radio handwheels - Use the configuration dialog to check the frequency spectrum - Change radio channel if required
230-04A7	**Error message** Other radio devices are disturbing radio handwheel operation **Cause of error** Other devices are disturbing radio handwheel operation **Error correction** - Use the configuration dialog to check the frequency spectrum - Change radio channel if required
230-04A8	**Error message** Could not activate or deactivate axis %2 **Cause of error** In a traverse range switchover, the position, speed and current controller of the axes to be deactivated must be switched off first. **Error correction** - Check the PLC program and adapt it if necessary. - Inform your service agency.
230-04A9	**Error message** Do not set the switch-off group, parameter set %2 **Cause of error** Setting the parameter CfgAxisHardware/MP_driveOffGroup is not permitted in controls with integrated functional safety (FS) from HEIDENHAIN. **Error correction** - Check the parameters. - The function must be implemented by the machine manufacturer via the SPLC program. - Inform your service agency.

Error number	Description
230-04AA	**Error message** The switch-off group must be set, parameter set %2 **Cause of error** The parameter CfgAxisHardware/MP_driveOffGroup must be set for controls without integrated functional safety (FS) from HEIDENHAIN. Refer to the Technical Manual for your control for the function and constraints of this parameter. **Error correction** - Check the parameters. - Inform your service agency.
230-04AB	**Error message** Parameter %2 not set for parameter set %3 **Cause of error** The parameter must be set for this parameter set. **Error correction** - Check the parameters. - Inform your service agency.
230-04AC	**Error message** Inputs for axis-group enabling are ignored **Cause of error** Inputs for axis-group enabling were found where the setting CfgHardware/MP_driveOffGroupInputs was defined. This machine ignores them. The safety functions STO, SBC, and SS1 can only be used via a PAE-H switch-off module. **Error correction** - Delete the setting CfgHardware/MP_driveOffGroupInputs. - Put the safety functions on terminals of a PAE module and configure them in accordance with the risk analysis of the machine. - Inform your service agency.
230-04AD	**Error message** PL inputs are not allowed as axis-group enabling **Cause of error** PL inputs for axis-group enabling were defined in the IOCP file. This machine ignores them. The safety functions STO, SBC, and SS1 can only be used via a PAE-H switch-off module. **Error correction** - Delete the axis-group enabling machine function from all PL input terminals. - Put the safety functions on terminals of a PAE module and configure them in accordance with the risk analysis of the machine. - Inform your service agency.

Error number	Description
230-04AE	**Error message** PAE module not configured in IOCP file **Cause of error** No machine functions for axis-group enabling were found in the IOCP file. The safety functions STO, SBC, and SS1 must be configured via a PAE-H switch-off module. The PAE module is presumably not configured. **Error correction** - Configure the PAE module correctly in the IOCP file. - Put the safety functions on terminals of a PAE module and configure them in accordance with the risk analysis of the machine. - Inform your service agency.
230-04AF	**Error message** Axis-group enabling machine function configured on PL module **Cause of error** The axis-group enabling machine function was defined on at least one terminal in the IOCP file even though the module is not a PAE module. The safety functions STO, SBC, and SS1 can only be used via a PAE-H switch-off module. **Error correction** - Put the safety functions on terminals of a PAE module and configure them in accordance with the risk analysis of the machine. - Inform your service agency.
230-04B0	**Error message** Invalid PWM frequency configured for axis %2 **Cause of error** An incorrect or invalid PWM frequency was configured. Frequencies up to 10 kHz are possible for CC 61xx and UEC 1xx. For Gen3 devices, depending on the UM device only frequencies up to 10 kHz are possible, or even only specific individual frequencies. **Error correction** Check and correct the configuration under CfgPowerStage->ampPwmFreq.

Error number	Description
230-04B3	**Error message** More safe axes are activated than are enabled in the SIK **Cause of error** The axis options in the SIK specify how many safe axes can be active at the same time. You have activated more safe axes over the machine configuration than are enabled as axis options in the SIK. **Error correction** - Check the machine configuration and correct it if necessary - If you need more safe axes, you can get a code number from HEIDENHAIN to enable them..
230-04B4	**Error message** SMC: Missing acknowledgment **Cause of error** SMC run-time error: - The safety-oriented software did not respond within the expected time period. - Generally high system load **Error correction** Check the system load
230-04B7	**Error message** Error in configuration of axis %2 **Cause of error** The axis is not configured correctly. **Error correction** Check the axis configuration in the following machine parameters and correct if required: - CfgSupplyModule/name - CfgPowerStage/ampPowerSupplyType - CfgPowerStage/ampBusVoltage - CfgPowerStage/supplyModule - CfgSupplyModule3xx/hsciUvIndex - CfgPowerStage/pwmPhaseShift - Inform your service agency
230-04B9	**Error message** The signal cannot be recorded **Cause of error** The necessary access right to record PLC signals is missing. **Error correction**

Error number	Description
230-04BA	**Error message** UVR%2 change of parameters requires restart. Shut down the control and restart it. **Cause of error** The UVR cannot apply the changed machine parameters without restarting. **Error correction** Restart the control
230-04BC	**Error message** Warning from EnDat speed encoder on motor in %1 axis ID: %2 SN: %3 Warning from EnDat position encoder in %1 axis ID: %2 SN: %3 **Cause of error** - Lower internal functional limit of the encoder has been exceeded. **Error correction** - Check the mounting of the encoder and correct it if necessary - Clean the encoder, if possible - Check whether any of the encoder specifications were not complied with, e.g. the supply voltage or ambient temperature. Ensure that the encoder is operated within the specifications. - Inform your service agency
230-04BD	**Error message** Advance warning from EnDat position encoder in %1 axis ID: %2 SN: %3 **Cause of error** - Internal functional limit of the encoder has been reached. The encoder can still be operated, but it is recommended that the encoder be checked. **Error correction** - Check the mounting of the encoder and correct it if necessary - Clean the encoder, if possible - Check whether any of the encoder specifications were not complied with, e.g. the supply voltage or ambient temperature. Ensure that the encoder is operated within the specifications. - Inform your service agency

Error number	Description
230-04BF	**Error message** Fatal error on inverter %2 Emergency stop by inverter %2 NC stop by inverter %2 **Cause of error** - Internal error in the inverter **Error correction** - Note any further messages - Inform your service agency
230-04C2	**Error message** Brake outputs can't be read by IOCP file **Cause of error** The IOCP file was not configured or cannot be read **Error correction** Inform your service agency
230-04C3	**Error message** Brake outputs are ignored **Cause of error** "Outputs for brake control" were defined in the IOCP file. These are not supported by the present machine configuration and will therefore be ignored. **Error correction** - Remove configured outputs for brake control from the IOCP file - Configure outputs for brake control through CfgBrake/MP_connection - Inform your service agency
230-04C4	**Error message** Two brake outputs refer to the same brake %1 **Cause of error** In the IOCP file, two "outputs for brake control" were configured with the same reference to one brake (CfgBrake). **Error correction** - Check the brake control configuration in the IOCP file and correct it - Inform your service agency

Error number	Description
230-04C5	**Error message** Control of brake %2 on axis %3 incorrectly configured **Cause of error** The configured output for brake control in the IOCP file refers to a hardware terminal other than the one currently being used by the axis. **Error correction** - Configure the output for brake control in the IOCP file to the correct terminal - Inform your service agency
230-04C6	**Error message** Brake %2: MP_connection must not be configured **Cause of error** The machine parameter CfgBrake/connection is not supported by the present machine configuration. **Error correction** Use IOconfig to configure "Output for controlling the brake" through the IOCP file
230-04C7	**Error message** Connection of brake %2 not configured **Cause of error** The output necessary for brake control is missing from the IOCP file **Error correction** - Configure the output for brake control for this brake in the IOCP file - Inform your service agency
230-04C8	**Error message** Braking connection for brake %2 not configured **Cause of error** Configuration of the brake connection is missing in CfgBrake/MP_connection **Error correction** - Configure the machine parameter CfgBrake/MP_connection - Inform your service agency

Error number	Description
230-04C9	**Error message** Acceptance testing of safety-relevant parameters %1 required **Cause of error** - The hardware configuration was significantly changed, e.g. different model of inverter - The configuration of the safety-related parameters was changed - The saved configuration was changed by the hardware defects **Error correction** - Check the safety-relevant configuration for the drive (parameter set) - Have suitable persons accept the configuration again, if necessary - Inform your service agency
230-04CA	**Error message** Braking connection for brake %2 incorrectly configured **Cause of error** An incorrect value was configured for the brake connection CfgBrake/MP_connection **Error correction** - Check the entry in the machine parameter CfgBrake/MP_connection and correct it as necessary - Inform your service agency
230-04CD	**Error message** Error during conversion of motor %1 **Cause of error** Converting the motor data from the old table 'PLC:\table\motor_oem.mot' to the table MOTOR_OEM failed. **Error correction** - Inform your service agency
230-04CE	**Error message** Communication with CC controller unit faulty **Cause of error** An error occurred during the internal communication with the CC controller unit. Internal information: error in acknowledging the SampleRate. The requested "blockSize" is not available. **Error correction**

Error number	Description
230-04CF	**Error message** A client with this ID is already logged on **Cause of error** The data interface to the configuration server is occupied. **Error correction** Inform your service agency.
230-04D0	**Error message** Configuration server is not ready **Cause of error** No readiness for communication through the data interface with the configuration server. **Error correction** Inform your service agency.
230-04D1	**Error message** The requested parameter is not available **Cause of error** A parameter that is not available was entered. **Error correction** - Check/Correct the entered parameter - If the error recurs, inform your service agency
230-04D2	**Error message** Faulty axis configuration (%2) **Cause of error** Axes that are not configured on the same CC controller unit are to be operated as torque-master-slaves. **Error correction** Check and correct the configuration
230-04D3	**Error message** Faulty monitoring of the software limit switches **Cause of error** Monitoring of the software limit switch was probably deactivated by the machine manufacturer via a macro. This is currently no longer permitted. **Error correction** - Contact the machine manufacturer - Information for machine manufacturers: check handling of the software limit switch in the OEM macro and correct as needed

Error number	Description
230-04D4	**Error message** Emergency stop by CC, %2 NC stop by CC, %2 **Cause of error** - Internal error in the controller unit **Error correction** - Note any further messages - Inform your service agency
230-04D6	**Error message** Temp. of the CC too high %2 **Cause of error** Temperature sensor detects an excessively high or low temperature within the housing of the controller unit. - Insufficient heat dissipation for the controller unit - Contaminated filter pads - Defective climate control unit in the electrical cabinet - Defective fan - Defective temperature sensor - Unfavorable mounting of components **Error correction** - Clean the filter pads - Check the climate control unit, and repair it if necessary - Replace the fan - Inform your service agency
230-04D7	**Error message** Error in acknowledging the sample rate of the UVR **Cause of error** The acknowledged sample rate of the UVR drive unit does not agree with the expected sample rate **Error correction** If the error recurs, inform your service agency.
230-04D8	**Error message** Error in acknowledging the sample rate of the UVR **Cause of error** The requested "blockSize" of the UVR drive unit is not available. **Error correction**

Error number	Description
230-04DD	**Error message** Input for SBC.GLOBAL configured incorrectly **Cause of error** For the use of the SBC.GLOBAL input terminal of the PAE module, the corresponding machine function was either not configured in the IO configuration (IOC file) or it was configured incorrectly. - No PAE module was configured. - A PAE module with an outdated version of the HDD file was configured. - More than one PAE module was configured. - SBC.GLOBAL was configured on the wrong module. **Error correction** - Check the IO configuration - Inform your service agency
230-04DE	**Error message** Input for SBC.GLOBAL must not be configured **Cause of error** Do not configure the machine function SBC.GLOBAL in the IO configuration (IOC file) of systems with integrated functional safety. **Error correction** - Check the IO configuration - Inform your service agency
230-04E0	**Error message** Progmmd. limit switch/protection zone incorrect for axis %2 **Cause of error** Protection zones for modulo axes should not be used with this version of the control software. **Error correction** - Remove protection zone for modulo axis - Do not configure the axis as a modulo axis
230-04E2	**Error message** Start position of axis (%2) not permissible for block scan **Cause of error** The starting position of the axis is outside of the permissible range. **Error correction** - Check the configuration of the software limit switches - Check the starting points of the axes in the NC program

Error number	Description
230-04E3	**Error message** No complete circle was recorded **Cause of error** During evaluation of the measured data, it was found that no complete circle was recorded. **Error correction** - Check whether the configured feed rate was achieved - Check the configured trigger speed
230-04E5	**Error message** Incorrect UVR/UEC configuration: %2, index invalid: %3 **Cause of error** No valid value entered for the machine parameter CfgSupplyModule3xx > hsciUvIndex. **Error correction** Check and correct the parameter: CfgSupplyModule3xx > hsciUvIndex
230-04E6	**Error message** Incorrect UVR/UEC configuration: config. duplicated: %1 - %2 **Cause of error** The parameter CfgSupplyModule3xx > hsciUvIndex is identical in the two stated instances of CfgSupplyModule3xx. That is not allowed. **Error correction** Check the parameter in all instances and correct as necessary: CfgSupplyModule3xx > hsciUvIndex
230-04E7	**Error message** Axis movement not permitted while switching the CLP filter **Cause of error** No axis may move while the CLP filter is being switched on or off. The movement may also result from superimpositioning (e.g., swing-frame grinding). Possible causes of the filter switching: - Switch-on or -off of TCPM (also M128 / M129) - Tilting the working plane **Error correction** Edit the NC program.

Error number	Description
230-04E8	**Error message** Underlying velocity is too low **Cause of error** There were changes to the algebraic sign over the course of the actual velocity. This reduces the quality of the determined characteristic values. **Error correction** - Increase the underlying velocity - The excitation amplitude might need to be reduced
230-04E9	**Error message** Not enough RAM to evaluate the measured data **Cause of error** Not enough RAM is available to evaluate the "Measure machine status" cycle. **Error correction** - Close any unnecessary applications - Restart the control
230-04EA	**Error message** MEASURE MACHINE STATUS: memory management faulty **Cause of error** Cannot delete an internal system file for temporary processing. **Error correction** Inform your service agency
230-04EC	**Error message** Axis-group enabling disabled while in motion **Cause of error** Axis-group enabling was disabled for safety reasons. **Error correction** If no reason can be identified, inform your machine tool builder

Error number	Description
230-04ED	**Error message** Parameters of current controller are not correct %2 **Cause of error** - The current controller (CfgCurrentControl) is not parameterized correctly Mixed parameterization is not permitted: Please use only (iCtrlPropGain and iCtrlIntGain) or (iCtrlPropGainD, iCtrlIntGainD, iCtrlPropGainQ, and iCtrlIntGainQ) **Error correction** - Correct the parameterization of the current controller Set (iCtrlPropGain = 0 and iCtrlIntGain = 0) or (iCtrlPropGainD = 0, iCtrlIntGainD = 0, iCtrlPropGainQ = 0, and iCtrlIntGainQ = 0)
230-04EE	**Error message** Parameters of current controller are not correct %2 **Cause of error** - The current controller (CfgCurrentControl) is not parameterized correctly iCtrlPropGain = 0, even though iCtrlIntGain > 0 **Error correction** - Correct the parameterization of the current controller: Set iCtrlPropGain > 0 or iCtrlIntGain = 0
230-04EF	**Error message** Parameters of current controller are not correct %2 **Cause of error** - The current controller (CfgCurrentControl) is not parameterized correctly iCtrlPropGainD = 0, even though iCtrlIntGainD > 0 **Error correction** - Correct the parameterization of the current controller: Set iCtrlPropGainD > 0 or iCtrlIntGainD = 0
230-04F0	**Error message** Parameters of current controller are not correct %2 **Cause of error** - The current controller (CfgCurrentControl) is not parameterized correctly iCtrlPropGainQ = 0, even though iCtrlIntGainQ > 0 **Error correction** - Correct the parameterization of the current controller: Set iCtrlPropGainQ > 0 or iCtrlIntGainQ = 0

Error number	Description
230-04F1	**Error message** Configuration error in Monitoring **Cause of error** Die Konfiguration des Monitorings (Komponenten- und Prozessüberwachung) ist mit dem beschriebenen Fehler fehlgeschlagen. **Error correction** Konfigurationsfehler durch Maschinenhersteller beheben lassen.
230-04F2	**Error message** Run-time error during calculation **Cause of error** Laufzeitfehler bei Berechnung des Monitorings **Error correction** Kundendienst benachrichtigen
230-04F3	**Error message** Error with external modules **Cause of error** In Bezug auf eine Monitoring-Funktion ist ein schwerwiegender Fehler mit externen Komponenten auf der Steuerung aufgetreten. **Error correction** Kundendienst benachrichtigen
230-04F4	**Error message** Internal (implementation) error **Cause of error** Schwerwiegender interner (Implementierungs-)Fehler bei Monitoring-Funktion. **Error correction** Kundendienst benachrichtigen
230-04F5	**Error message** Missing rights **Cause of error** Rechte für die auszuführende Aktion fehlen **Error correction** Kundendienst benachrichtigen

Error number	Description
230-04F6	**Error message** Monitoring task results in a warning **Cause of error** Auszuführende Aktion im Montioring löst eine Warnung aus **Error correction** Kundendienst benachrichtigen
230-04F7	**Error message** Error while setting up Process Monitoring **Cause of error** Beim Einrichten der Prozessüberwachung ist ein Fehler aufgetreten: "Monitoring Meta Data"-Datei fehlerhaft **Error correction** Kundendienst benachrichtigen
230-04F8	**Error message** Faulty configuration of Multicast data **Cause of error** Die Konfiguration der Beauftragung der Multicast-Daten ist mit dem beschriebenen Fehler fehlgeschlagen. **Error correction** Konfigurationsfehler durch Maschinenhersteller beheben lassen.
230-04F9	**Error message** Internal error in the Multicast data interface **Cause of error** Schwerwiegender interner (Implementierungs-)Fehler in der Multicast-Datenschnittstelle. **Error correction** Kundendienst benachrichtigen
230-04FA	**Error message** Realtime Container error in the Multicast data interface **Cause of error** Realtime Container Fehler in der Multicast-Datenschnittstelle aufgetreten. **Error correction** Kundendienst benachrichtigen.

Error number	Description
230-04FB	**Error message** Realtime Container error in monitoring tasks **Cause of error** Realtime Container Fehler im Monitoring aufgetreten. **Error correction** Kundendienst benachrichtigen.
230-04FC	**Error message** Error while calculating the indicators **Cause of error** Fehler im Zusammenhang mit den Indikatoren im Monitoring aufgetreten. **Error correction** Kundendienst benachrichtigen.
230-04FD	**Error message** Internal error in the NC reactions for monitoring tasks **Cause of error** Fehler bei den Reaktionen im Monitoring aufgetreten. **Error correction** Kundendienst benachrichtigen.
230-04FE	**Error message** Internal error in monitoring tasks **Cause of error** Schwerwiegender interner (Implementierungs-)Fehler im Monitoring. **Error correction** Kundendienst benachrichtigen.
230-04FF	**Error message** Error in the table server of monitoring tasks **Cause of error** Es sind Fehler aufgetreten, die Tabellen im Monitoring betreffen. **Error correction** Kundendienst benachrichtigen.
230-0500	**Error message** Error in the formulas used for monitoring tasks **Cause of error** Es sind Fehler in Bezug auf die im Monitoring verwendeten Formeln aufgetreten. **Error correction** Kundendienst benachrichtigen.

Error number	Description
230-0501	**Error message** Faulty configuration of the monitoring tasks **Cause of error** An error occurred while configuring a monitor. **Error correction** Inform your service agency.
230-0502	**Error message** Software option for component monitoring is not enabled **Cause of error** Die Komponentenüberwachung kann nicht verwendet werden. Die notwendige Software-Option ist nicht freigeschaltet. **Error correction** - Software-Option für die Komponentenüberwachung im SIK freischalten - Kundendienst benachrichtigen
230-0503	**Error message** Process Monitoring software option is missing **Cause of error** The software option for process monitoring is not enabled. Process monitoring cannot be used. **Error correction** Inform your service agency
230-0504	**Error message** Cancel program after NC stop because of error %2 **Cause of error** Program execution was canceled after an NC stop. Reason: The CC controller triggered an error. **Error correction** Note the information on remedies while the CC error is displayed.
230-0505	**Error message** Frequency too high for eval. slot %s in envelope curve spectrum **Cause of error** The frequency to be evaluated is beyond the Nyquist frequency. **Error correction** - Select an adequate frequency below the Nyquist frequency

Error number	Description
230-0506	**Error message** No monitoring due to an upstream error **Cause of error** Due to an upstream error, Component Monitoring and Process Monitoring were deactivated. **Error correction** Inform your service agency.
231-4003	**Error message** 4003 EMERGENCY STOP is active (emergency stop test) **Cause of error** **Error correction**
231-4004	**Error message** 4004 EMERGENCY STOP is inactive (emergency stop test) **Cause of error** **Error correction**
231-4005	**Error message** 4005 Stack overflow warning **Cause of error** **Error correction**
231-4007	**Error message** 4007 No interrupt from the gate array **Cause of error** **Error correction**
231-4008	**Error message** 4008 Answer to a host command was too late **Cause of error** **Error correction**
231-4009	**Error message** 4009 False interrupt (AC failure, emergence stop) **Cause of error** **Error correction**
231-400B	**Error message** 400B Host is prompted to synchronize **Cause of error** **Error correction**

Error number	Description
231-4011	**Error message** 4011 Axis %1: measurement interrupted by NC Stop **Cause of error** The cycle was interrupted by an NC Stop while measuring. The measurement was canceled and the data discarded. **Error correction** - Continue the cycle with NC Start - Repeat the measurement
231-4110	**Error message** 4110 Encoder contamination in axis %1 **Cause of error** **Error correction**
231-4120	**Error message** 4120 Excessive frequency in encoder for axis %1 **Cause of error** **Error correction**
231-4130	**Error message** 4130 Contamination of Z1 track in encoder for axis %1 **Cause of error** **Error correction**
231-4140	**Error message** 4140 Power module warning: Heat sink temperature in axis %1 **Cause of error** **Error correction**
231-4150	**Error message** 4150 Power module warnung for axis %1 **Cause of error** **Error correction**
231-4160	**Error message** 4160 Gate array status reg. = not nicht stored (axis %1) **Cause of error** **Error correction**

Error number	Description
231-4170	**Error message** 4170 Error in temperature measurement for axis %1 **Cause of error** **Error correction**
231-4200	**Error message** 4200 PLC: Drive is ready for operation in axis %1 **Cause of error** **Error correction**
231-4210	**Error message** 4210 PLC: Drive is not ready for operation in axis %1 **Cause of error** **Error correction**
231-4220	**Error message** 4220 Standstill recognition (V=0 with IQ_max): (axis %1) **Cause of error** **Error correction**
231-4230	**Error message** 4230 End of standstill recognition in axis %1 **Cause of error** **Error correction**
231-4240	**Error message** 4240 Warning for I^2*t monitoring in axis %1 **Cause of error** **Error correction**
231-4250	**Error message** 4250 End of warning for I^2*t monitoring in axis %1 **Cause of error** **Error correction**
231-4260	**Error message** 4260 Limit through I^2*t monitoring **Cause of error** **Error correction**

Error number	Description
231-4270	**Error message** 4270 End of limit through I^2*t monitoring in axis %1 **Cause of error** **Error correction**
231-4280	**Error message** 4280 Error during REF scan in axis %1 **Cause of error** **Error correction**
231-4290	**Error message** 4290 Drive switch-off (inactive RDY signal) in axis %1 **Cause of error** **Error correction**
231-4300	**Error message** 4300 Drive enable in axis %1 **Cause of error** **Error correction**
231-4310	**Error message** 4310 Drive disabled, e.g. through emergency stop in axis %1 **Cause of error** **Error correction**
231-4400	**Error message** 4400 Excessive following error (axis: %1) **Cause of error** - The following error of an axis exceeds one of the following error limits. - The acceleration entered is too large. - The motor does not move in spite of “Drive on.” **Error correction** - Reduce the contouring feed rate, increase the rotational speed. Remove any possible sources of vibration. Inform your service agency if the error occurs frequently. - Check the acceleration (CfgFeedLimits/maxAcceleration) - Check the following error limits (CfgPosControl/servoLag-Min[1/2] or servoLagMax[1/2]) - The motor current must not be under limitation during acceleration

Error number	Description
231-5100	**Error message** 5100 Endless loop: Time monitoring (value: %1) **Cause of error** **Error correction**
231-5200	**Error message** 5200 Compensated angle error alignment (value: %1) **Cause of error** **Error correction**
231-5300	**Error message** 5300 Timeout of shaft speed interrupt (value: %1) **Cause of error** **Error correction**
231-5500	**Error message** 5500 Emergency stop failure **Cause of error** **Error correction**
231-6002	**Error message** 6002 Status of machine-on input = 1 after "hcSgMaschineEin" **Cause of error** **Error correction**
231-6003	**Error message** 6003 Status of machine-on input = 0 after "hcSgMaschineEin" **Cause of error** **Error correction**
231-6005	**Error message** 6005 STOP1 release due to error in T2 test **Cause of error** **Error correction**
231-6006	**Error message** 6006 Tool changer: "SHS2"-output change to open chuck **Cause of error** **Error correction**

Error number	Description
231-6016	**Error message** 6016 STOP2 release: Incorrect power supply **Cause of error** **Error correction**
231-6017	**Error message** 6017 STOP2 release: Out of temperature range **Cause of error** **Error correction**
231-6018	**Error message** 6018 Request for test of pulse deletion **Cause of error** **Error correction**
231-6100	**Error message** 6100 Cut-out channel test: Status change -STO.A.x %1 = 0->1 **Cause of error** **Error correction**
231-6140	**Error message** 6140 Negative position monitoring with stop 1 **Cause of error** **Error correction**
231-8000	**Error message** 8000 CC%2 Warning: Temperature of controller card is high **Cause of error** - The temperature of the controller card has exceeded a warning threshold - Ambient temperature is too high **Error correction** - Check the air conditioning of the electrical cabinet - Check whether the fan is working - Inform your service agency
231-8001	**Error message** 8001 CC%2 Warning: Temperature of controller card is low **Cause of error** - The temperature of the controller card has fallen below a warning threshold - Ambient temperature is too low **Error correction** - Check the air conditioning of the electrical cabinet - Inform your service agency

Error number	Description
231-8002	**Error message** 8002 CC%2 warning: Fan speed is low **Cause of error** - The rotational speed of the fan in the CC controller unit has exceeded a warning threshold **Error correction** - Check the fan - Inform your service agency
231-8003	**Error message** 8003 CC%2 early warning: dc-link current too high **Cause of error** - The DC-link voltage of the temperature of the supply module has exceeded a warning threshold **Error correction** - Reduce the DC-link voltage by: - Adjust the acceleration/braking ramps of the axes/spindles - Check the "motPbrMax" and "motPMa" machine parameters - Inform your service agency
231-8004	**Error message** 8004 CC test software loaded **Cause of error** - In the control there is a non-released CC test program without valid CRC program checksum - No valid CRC checksum value was entered in the binary file of the CC software - The MC software version does not support the CRC checksum comparison **Error correction** - The software can be used after the error acknowledgment for initial servicing or error diagnostics - Check the software version - Read out the log - Inform your service agency

Error number	Description
231-8005	**Error message** 8005 Oscilloscope signal not supported **Cause of error** - At least one of the selected signals of the internal oscilloscope is not supported with this controller unit (CC422) - The following signals are not supported by the controller unit: P mech., P elec., DSP debug, DC-link P, CC DIAG, I actual, actl. Id, Iq max, U noml, int. diag., motor A and motor B **Error correction** - Please do not select the following signals: - P mech. - P elec. - DSP debug - DC-link P - CC DIAG - I actual - Actl. Id - U nominal - int. Diag. - Motor A - Motor B - Max. Iq
231-8010	**Error message** 8010 LSV2 transmission error **Cause of error** - Error in data transfer by LSV2 protocol **Error correction** - Press the CE key to acknowledge the error. - Error does not impair the control functions. - Inform your service agency.
231-8040	**Error message** 8040 Heat sink temp. in UV 1xx **Cause of error** - Heat-sink temperature of UV 1xx power supply unit too high - Temperature in the electrical cabinet too high - Contaminated filter pads - Defective climate control unit in the electrical cabinet - Defective fan in the UV - Defective temperature sensor - If the heat-sink temperature continues to increase, the unit will be switched off **Error correction** - Stop the machine and let it cool down - Continue working with lower power (reduce the feed rate) - Clean the filter pads - Check the climate control unit in the electrical cabinet - Inform your service agency

Error number	Description
231-8041	**Error message** 8041 Excessive Iz in UV 1xx **Cause of error** - DC-link current of UV 1xx power supply unit too high - Overload of the machine while machining a workpiece **Error correction** - Continue working with lower power (reduce feed rate, replace worn tool, etc.)
231-8060	**Error message** 8060 Leakage current in UV 1xx **Cause of error** - Insulation problem (e.g. defective motor, contamination of the inverter, humidity) - Check the motor connection for a ground fault **Error correction** - Isolate the cause by deselecting specific axes - Replace the motor of the affected axis or check for a ground fault - Replace the power cable of the affected axis or check for a ground fault - Replace the inverter of the affected axis or check for a ground fault - Check the motors of all selected axes - Inform your service agency
231-8061	**Error message** 8061 Power module not ready **Cause of error** - Readiness signal of the power supply module is inactive after the servo control starts. - Master contactor has opened - Error in PLC program - Power supply module, inverter defective **Error correction** - Try restarting the inverter. If the error recurs: - Inform your service agency - Check the wiring (main contactor) - Check the PLC program - Exchange the power supply module

Error number	Description
231-8062	**Error message** 8062 Limit: Uz too small **Cause of error** - The dc-link voltage from the supply unit fell below the defined limit. **Error correction** - Inform your service agency - Failure of the power supply at the supply module - Excessive power consumption by the axes->activate the power limitation of the spindle
231-8063	**Error message** 8063 CC%2 timeout in S status test **Cause of error** - The MC activates the S status evaluation too late after a test. - The maximum test switch-off time was exceeded. **Error correction** - Inform your service agency
231-8064	**Error message** 8064 AC powerfail AC powerfail **Cause of error** An error occurred during the AC Fail process. The line voltage was intermittently interrupted. Possible causes: - Power failure - Dropout in line power - Defective protection of the line power supply - Check the wiring of the line power supply **Error correction** - Check the line power circuit breakers - Check the power supply wiring - Check the quality of the line voltage (possible dropouts) - Inform your service agency

Error number	Description
231-8065	**Error message** 8065 DC powerfail DC powerfail 806C DC powerfail **Cause of error** An error occurred during the DC Fail process. The dc-link voltage is below the specified limit. **Error correction** - Check the dc-link voltage - Check the DC-link charging contactor for interruptions - Check the line power supply - Check the line power circuit breakers - Check the wiring of the line power - Check the quality of the line voltage (possible dropouts) - Inform your service agency
231-8066	**Error message** 8066 CC%2 maximum temperature of controller card exceeded 8042 CC%2 maximum temperature of controller card exceeded **Cause of error** - The maximum temperature of the controller card (CC) was exceeded - Ambient temperature is too high **Error correction** - Check the air conditioning of the electrical cabinet - Check whether the fan is working - Inform your service agency
231-8067	**Error message** 8067 CC%2 temperature of controller card below minimum 8043 CC%2 temperature of controller card below minimum **Cause of error** - The temperature of the controller card (CC) is below the minimum - Ambient temperature is too low **Error correction** - Check the air conditioning of the electrical cabinet - Inform your service agency

Error number	Description
231-8068	**Error message** 8068 CC%2 Fan speed too low 8044 CC%2 Fan speed too low **Cause of error** - The rotational speed of the fan in the CC controller unit has exceeded a monitoring threshold **Error correction** - Check the fan - Exchange the controller unit - Inform your service agency
231-8069	**Error message** 8069 IGBT error in the supply module **Cause of error** - The supply module switched off with an IGBT error (overload). **Error correction** - Check the power consumption of axes and spindle - Exchange the power supply module - Inform your service agency
231-806A	**Error message** 806A Unknown SPI expansion module **Cause of error** - The SPI plug-in module (for CC or MC) is unknown or provides an unknown code. - The SPI plug-in module is defective. **Error correction** - Remove the defective SPI plug-in module. - Exchange the defective SPI plug-in module. - Inform your service agency.
231-806B	**Error message** 806B Powerfail AC drive cntrller **Cause of error** An "AC fail" occurred during operation. This means that the line power supply was temporarily interrupted. Possible causes: - Power failure - Dropout in line power - Defective protection of the line power supply - Defective wiring of the line power supply **Error correction**

Error number	Description
231-8080	**Error message** 8080 Uz UV 1xx exceeds max. **Cause of error** - DC-link voltage of the power supply unit too high - Defective braking resistor - Defective power supply unit (infeed/regenerative feedback module) - Interruption in the primary supply (fuses, wires, etc.) - Energy recovery not possible **Error correction** - Inform your service agency - Check the configuration datum (braking of the spindle) - Check the braking resistor - Replace the power supply unit - Check the fuses and wiring of the primary supply
231-8081	**Error message** 8081 Uz is too small **Cause of error** - DC-link voltage of the supply unit is too low **Error correction** - Inform your service agency - Failure of the power supply at the supply module - Excessive power consumption by the axes->activate the power limitation of the spindle - Check MP2192
231-8082	**Error message** 8082 MC shut-off signal "-STO.A.MC.WD" is active **Cause of error** - The shut-off signal "-STO.A.MC.WD" of the MC is active - Check the entry in machine parameter CfgCycleTime -> watchdogTime - Hardware defective - The drive was switched off due to an internal error of the MC **Error correction** - Inform your service agency
231-8086	**Error message** 8086 Probing already active **Cause of error** Internal software error **Error correction** - Inform your service agency - Check the software version

Error number	Description
231-8092	**Error message** 8092 Pos. contr. cyc. time error **Cause of error** - MC is providing erroneous cycle time for CC position controller - Hardware error **Error correction** - Inform your service agency - Check the configuration datum ipoCycle - Exchange the drive control board
231-8093	**Error message** 8093 CC%2 HSCI communication code=%4 address=%5 **Cause of error** - The HSCI communication monitor reports a transmission error - Hardware of an HSCI participant is defective **Error correction** - Check the HSCI cabling - Exchange the defective hardware - Inform your service agency

Error number	Description
231-8093	**Error message** 8093 CC%2 HSCI data package not received Code=%4 Address=%5 8093 CC%2 HSCI data package not received Code=%4 Address=%5 8093 CC%2 HSCI data package not received Code=%4 Address=%5 8093 CC%2 HSCI data packet not received Code=%4 Address=%5 **Cause of error** There was an error in HSCI communication. An expected data packet with the above described HSCI address could not be received. Code 2101: Missing cyclic data 2102: Missing asynchronous data 2103: Missing asynchronous data 2 2104: Missing cyclic data of low priority - Sporadic interruption of the HSCI communication through contact problems (connector) or external EMC disturbance - Grounding problems in the HSCI system - Disturbance in the power supply of an HSCI participant - Power supply fluctuations are too low or too high Power supply of an HSCI participant - Defect of an HSCI participant **Error correction** - Check the cabling (HSCI connection) - Check the ground and power supply of the HSCI participant - Check the software version. - Inform your service agency.

Error number	Description
231-8093	**Error message** 8093 CC%2 HSCI: Wrong ID data from HSCI device Code=%4 Address=%5 8093 CC%2 HSCI: Wrong ID data from HSCI device Code=%4 Address=%5 8093 CC%2 HSCI: Wrong ID data from HSCI device Code=%4 Address=%5 8093 CC%2 HSCI: Wrong ID data from HSCI device Code=%4 Address=%5 8093 CC%2 HSCI: Wrong ID data from HSCI device Code=%4 Address=%5 8093 CC%2 HSCI: Wrong ID data from HSCI device Code=%4 Address=%5 **Cause of error** There was an error in HSCI communication. an HSCI device with functional safety FS sent an incorrect identifica-tion data to the HSCI device with the above described HSCI address. Code 2201: Missing cyclic telegram 2202: Data telegram contains incorrect CRC checksum 2203: Data telegram contains incorrect watchdog counter 2204: Data telegram indicates incorrect channel 2205: Data telegram contains incorrect HSCI address 2206: Data telegram contains incorrect number of inputs - Incorrect HSCI configuration (IOC file) or wiring error - Defect of an HSCI participant
231-8093	**Error message** 8093 CC%2 Error in HSCI telegram between MC-CC Code=%4 Address=%5 8093 CC%2 Error in HSCI telegram between MC-CC Code=%4 Address=%5 8093 CC%2 Error in HSCI telegram between MC-CC Code=%4 Address=%5 **Cause of error** There was an error in HSCI communication. An expected data packet with the above described HSCI address could not be received. Code 2301: Missing telegram from MC to controller unit 2302: Data telegram contains incorrect CRC checksum 2303: Data telegram contains incorrect watchdog counter - Sporadic problems of the HSCI connection or external disturbance - Defect of an HSCI participant **Error correction** - Check the cabling (HSCI connection) - Check the software version. - Inform your service agency.

Error number	Description
231-8093	**Error message** 8093 CC%2 Error in HSCI telegram between CCs Code=%4 Address=%5 8093 CC%2 Error in HSCI telegram between CCs Code=%4 Address=%5 8093 CC%2 Error in HSCI telegram between CCs Code=%4 Address=%5 8093 CC%2 Error in HSCI telegram between CCs Code=%4 Address=%5 **Cause of error** An error occurred in the HSCI communication between two or more CC 61xx or UEC 1xx controller units. Code 2401: Missing transmission telegram between the controller units 2402: Data telegram contains incorrect CRC checksum 2403: Data telegram contains incorrect watchdog counter 2404: Incorrect index of a controller unit - Sporadic interruption of the HSCI communication through contact problems (connector) or external disturbance - Defect of an HSCI participant **Error correction** - Check the cabling (HSCI connection) - Check the software version. - Inform your service agency.
231-8093	**Error message** 8093 CC%2 Configuration error of an HSCI device with FS Code=%4 Address=%5 8093 CC%2 Configuration error of an HSCI device with FS Code=%4 Address=%5 8093 CC%2 Configuration error of an HSCI device with FS Code=%4 Address=%5 8093 CC%2 Configuration error of an HSCI device with FS Code=%4 Address=%5 8093 CC%2 Configuration error of an HSCI device with FS Code=%4 Address=%5 8093 CC%2 Configuration error of an HSCI device with FS Code=%4 Address=%5 8093 CC%2 Configuration error of an HSCI device with FS Code=%4 Address=%5 **Cause of error**

Error number	Description
231-8093	**Error message** 8093 CC%2 Alarm message from HSCI device Code=%4 Address=%5 8093 CC%2 Alarm message from HSCI device Code=%4 Address=%5 8093 CC%2 Alarm message from HSCI device Code=%4 Address=%5 8093 CC%2 Alarm message from HSCI device Code=%4 Address=%5 8093 CC%2 Alarm message from HSCI device Code=%4 Address=%5 8093 CC%2 Alarm message from HSCI device Code=%4 Address=%5 8093 CC%2 Alarm message from HSCI device Code=%4 Address=%5 8093 CC%2 Alarm message from HSCI device Code=%4 Address=%5 8093 CC%2 Alarm message from HSCI device Code=%4 Address=%5
231-8094	**Error message** 8094 CC%2 HSCI sampling interval not transferred **Cause of error** - Internal software error **Error correction** - Inform your service agency
231-8130	**Error message** 8130 Motor brake defective %1 **Cause of error** - Motor brake defective **Error correction** - Inform your service agency - Check the motor brake control - Exchange the motor
231-8150	**Error message** 8150 Field orient. successful %1 8690 Field orientation successful %1 **Cause of error** - Field orientation successful **Error correction** - Press CE to acknowledge the message

Error number	Description
231-8160	**Error message** 8160 Actual current value too high %1 8630 Actual current value too high %1 **Cause of error** - The maximum permissible current of the power stage was exceeded **Error correction** - Check the current controller adjustment - Inform your service agency - Check the motor table, power stage table and configuration data - Check the system for short circuits
231-8190	**Error message** 8190 Error in TNCopt measurement %1 **Cause of error** - The TNCopt measurement was cancelled **Error correction** - Check the TNCopt version - Check whether the control is active - Inform your service agency
231-81A0	**Error message** 81A0 Invalid diagnostics channel %1 **Cause of error** - Invalid CC-diag channel is selected in the oscilloscope. **Error correction** - Select another channel
231-81A1	**Error message** 81A1 CC %2 axis %1: Invalid DSP debug channel %4 **Cause of error** - Invalid DSP debug signal selected in the oscilloscope **Error correction** - Select another signal
231-81A2	**Error message** 81A2 Axis %1: Quantity of channels per inverter exceeded 81A3 Axis %1: Quantity of channels per encoder exceeded **Cause of error** The maximum permissible number of oscilloscope channels per device was exceeded. **Error correction** Reduce the number of oscilloscope channels for the device.

Error number	Description
231-81A4	**Error message** 81A4 Axis %1: result of weighing was deleted **Cause of error** The result of the weighing run was deleted. Cycle 239 was not deactivated. **Error correction** Deactivate Cycle 239 before switching to another setting.
231-81B0	**Error message** 81B0 DQ com error of inverter %1 Po=%4 Dev=%5 Error=%6 **Cause of error** - DRIVE-CLiQ communication to the inverter of the named axis is disturbed - DRIVE-CLiQ communication at the named output (Po=Port) has been interrupted - Error code (error) of the DSA link (third additional info): - Output in decimal, to be interpreted in binary, consisting of 8 bits: Bit(s) Meaning 7 Interrupt triggered 6..5 Error group: "00" Telegram reception error "01" Other reception error "10" Transmission error "11" Other errors 4 Telegram too early 3..0 More detailed information: 0x1: CRC error 0x2: Telegram too short 0x3: Telegram too long 0x4: Length byte incorrect 0x5: Wrong telegram type 0x6: Incorrect address 0x7: No SYNC telegram 0x8: Unexpected SYNC telegram 0x9: ALARM bit received 0xA: Life sign is missing 0xB: Synchronization error of the alternative cyclic data traffic **Error correction** - Inspect the DRIVE-CLiQ cabling - Exchange the inverter - Exchange the CC controller unit. - Inform your service agency.

Error number	Description
231-81C0	**Error message** 81C0 DQ com error motor encdr %1 Po=%4 Dev=%5 Error= %6 **Cause of error** - DRIVE-CLiQ communication to the speed encoder (motor encoder) of the named axis is disturbed. - DRIVE-CLiQ communication at the named output (Po=Port) has been interrupted. - Error code (error) of the DSA link (third additional info): - Output in decimal, to be interpreted in binary, consisting of 8 bits: Bit(s) Meaning 7 Interrupt triggered 6..5 Error group: "00" Telegram reception error "01" Other reception error "10" Transmission error "11" Other errors 4 Telegram too early 3..0 More detailed information: 0x1: CRC error 0x2: Telegram to short 0x3: Telegram to long 0x4: Length byte incorrect 0x5: Wrong telegram type 0x6: Incorrect address 0x7: No SYNC telegram 0x8: Unexpected SYNC telegram 0x9: ALARM bit received 0xA: Life sign is missing 0xB: Synchronization error of the alternative cyclic data traffic **Error correction** - Inspect the DRIVE-CLiQ cabling - Check the encoder connection - Exchange the speed encoder - Exchange the CC controller unit. - Inform your service agency.

Error number	Description
231-8300	**Error message** 8300 Motor brake defective %1 **Cause of error** - Motor brake defective **Error correction** Warning: Hanging axes cannot be supported under certain circumstances. The axis can fall down. Do not enter the area of danger under the axis! - Move the axis to a safe position before power-off - Inform your service agency - Check controls for motor brakes - Exchange the motor
231-8310	**Error message** 8310 No current in brake test %1 **Cause of error** - Motor is not properly connected - Inverter is incorrectly connected - Inverter is defective - Motor is defective **Error correction** - Inform your service agency - Check the wiring of the motor and inverter - Check the inverter - Check the motor
231-8320	**Error message** 8320 PIC: Actual value does not equal the nominal value %1 **Cause of error** - Check the entered power stage in the machine parameter file (MP_ampName) - Check the nominal value of the PICS (value in column S for the selected power stage of the file inverter.inv) - If necessary, change the PWM frequency (to >= 5 kHz) - If necessary, replace the power module **Error correction** - Nominal value (value in column S for the selected power stage of the file inverter.inv) = 1, but: - There is no PIC - PIC cannot be switched - PIC access has been deactivated (MP_motEncCheckOff, Bit16=1) - Nominal value (value in column S of the selected power stage of the file inverter.inv) = 0 however: - PIC access has been deactivated (MP_motEncCheckOff, bit 16=1) and PIC was already switched to 1

Error number	Description
231-8330	**Error message** 8330 Brake test was canceled %1 **Cause of error** The brake test was canceled by - PLC through Module 9161 = 0 - Missing enabling signal (emergency stop, X150, ...) - Other error message **Error correction** - Check the PLC program - Check the enabling signals - Inform your service agency
231-8420	**Error message** 8420 Excessive temperature of power module %2.s **Cause of error** - Axis is being loaded too heavily (overload, temperature) - Insufficient cooling in the electrical cabinet **Error correction** - Reduce the load – Check the climate control in the electrical cabinet. - Inform your service agency
231-8430	**Error message** 8430 Error during axis change %1 **Cause of error** - An axis was removed from the machine configuration while still in the closed control loop **Error correction** - Check the PLC program and edit if necessary - Check the configuration datum axisMode - Inform your service agency
231-8440	**Error message** 8440 Field orient. successful %1 **Cause of error** - Field orientation successfully completed. **Error correction** - Acknowledge the message with CE.
231-8600	**Error message** 8600 No drive-on command for %1 **Cause of error** - Speed controller is waiting for the "drive on" command; the PLC program has sent no "drive on" command. **Error correction** - Check the PLC program. - Inform your service agency. - Check the software version.

Error number	Description
231-8610	**Error message** 8610 I2T value is too high %1 **Cause of error** - The load of the drive is too high for the duration **Error correction** - Reduce the load or the duration - Inform your service agency - Check the motor table, power stage table and configuration data - Check whether the motor and power module are designed for the load
231-8620	**Error message** 8620 Load is too high %1 **Cause of error** - Drive has maximum current and cannot accelerate - Excessive load (torque, power) on the drive **Error correction** - Reduce the load on the drive - Inform your service agency - Check the motor table, power stage table and configuration data - Check whether the motor and power module are designed for the load
231-8640	**Error message** 8640 I2T value of motor is too high %1 **Cause of error** - The load of the motor is too high over the duration **Error correction** - Reduce the load or the duration - Check the motor table and configuration datum - Check whether the motor is designed for the load - Inform your service agency
231-8650	**Error message** 8650 I2T value of power module is too high %1 **Cause of error** - The load of the power module is too high over the duration **Error correction** - Reduce the load or the duration - Check the power module and configuration datum - Check whether the power module is designed for the load - Inform your service agency

Error number	Description
231-8680	**Error message** 8680 DQ inverter %1: Maximum current limiting **Cause of error** - The maximum current read from the inverter is smaller than the maximum current in the inverter table. - At a PWM frequency > 4 kHz: The output current has been derated too far. - The permissible load application of the inverter was exceeded. **Error correction** - Reduce the PWM frequency - Reduce the application load of the inverter - Inform your service agency
231-8800	**Error message** 8800 Signal LT-RDY inactive %1 8810 Signal LT-RDY inactive %1 **Cause of error** - Undesirable inverter switch-off during control of a vertical axis (caused by vertical axis). **Error correction** - Inform your service agency. - Check the PLC program. - Check the wiring of the inverter.
231-8820	**Error message** 8820 Field angle unknown %1 **Cause of error** - The field angle of the motor on the encoder reference point has not yet been ascertained. **Error correction** - Inform your service agency - Check the entry in the configuration datum motFieldAdjust-Move - If required, find field angle in the "current controller adjust-ment" mode (press the "FIELD ORIENT." soft key) Caution: The motor must be freely rotatable (no clamping, no hanging axis, no mechanical constraints). - Check the "type of encoder" entry in the motor table

Error number	Description
231-8830	**Error message** 8830 EnDat: no field angle %1 **Cause of error** - The field angle of a motor with unaligned EnDat encoder has not yet been ascertained- The transferred EnDat serial number does match the one saved **Error correction** - Inform your service agency - If required, find field angle in the "current controller adjustment" mode (press the "FIELD ORIENT." soft key) - "Type of encoder" entry in the motor table - Check the configuration datum motEncType
231-8840	**Error message** 8840 Axis not available %1 **Cause of error** - Starting command for unavailable axis. **Error correction** - Inform your service agency. - Check the software version.
231-8850	**Error message** 8850 Drive still active %1 **Cause of error** - Position measurement (Z1 track) was started although the drive is still active. **Error correction** - Inform your service agency. - Check the software version.
231-8860	**Error message** 8860 Input frequency of speed encoder %1 **Cause of error** - Noise on speed encoder signals - Signal connector: Poor contact or penetration of humidity - Humidity has entered the motor **Error correction** - Inform your service agency - Check the encoder signals - Check the shielding

Error number	Description
231-8870	**Error message** 8870 Input frequency of position encoder %1 **Cause of error** - Noise on position encoder signals - Penetration of humidity **Error correction** - Inform your service agency - Check the encoder signals - Check the shielding
231-8880	**Error message** 8880 No enabling while field angle %1 is being found **Cause of error** - Enabling is canceled while the field angle is being determined (e.g. PLC program, emergency stop, X150/X151, monitoring function). **Error correction** - Inform your service agency - Check the PLC program (most frequent cause)
231-8890	**Error message** 8890 TRC: Wrong type of motor %1 **Cause of error** - The axis for which the torque ripple compensation was activated through MP2260.x is not driven by a synchronous or linear motor. **Error correction** - Inform your service agency - To deactivate the compensation, delete the entry in MP2260.x
231-88A0	**Error message** 88A0 TRC: Wrong control %1 **Cause of error** - The compensation file was generated for a different control from this. Using a compensation file copied from another control is not allowed. **Error correction** - Inform your service agency - Find again the compensation parameters with TNCopt under Optimization/Torque Ripple Compensation - To deactivate the compensation, delete the entry in MP2260.x

Error number	Description
231-88B0	**Error message** 88B0 TRC: Wrong file – motor %1 **Cause of error** - The compensation file was generated for a motor with a different speed encoder with EnDat interface from this one. Using a compensation file copied from another control is not allowed. **Error correction** - Inform your service agency - Find again the compensation parameters with TNCopt under Optimization/Torque Ripple Compensation - To deactivate the compensation, delete the entry in MP2260.x
231-88C0	**Error message** 88C0 Max. nominal motor speed %1 exceeded **Cause of error** - Axis: Maximum feed rate is greater than the maximum motor speed (N-MAX) multiplied by the configuration datum distPerMotorTurn - Spindle: Maximum spindle speed is greater than the maximum motor speed (N-MAX) multiplied by the gear transmission ratio - The relationship between the line count of the position encoder and that of the motor encoder is faulty - The N-MAX entry in the motor table is faulty - Incorrect entry in the configuration datum motName - EcoDyn: The selected feed rate exceeds the max. permissible voltage **Error correction** - Inform your service agency - Check the N-MAX entry in the motor table - Check the configuration data maxFeed, manualFeed and distPerMotorTurn - Check the configuration data for spindle speed - Check the STR column in the motor table and line count in the spindle parameter block (configuration datum "posEncoderIncr") - Check all configuration data under CfgServoMotor

Error number	Description
231-88D0	**Error message** 88D0 Kinematic compensation %1 not possible **Cause of error** Kinematic compensation via compensation file is possible only for - Double-speed axes - PWM frequencies less than or equal to 5 kHz **Error correction** - Check the machine parameter: - Check the entry in SelAxType. - Check the entry in AmpPwmFreq. - Check the compensation file. - Inform your service agency.
231-88E0	**Error message** 88E0 Brake test %1 not possible **Cause of error** - A brake test is not possible because the axis is not in a control loop. **Error correction** - Inform your service agency - Software error
231-88F0	**Error message** 88F0 Inverter and supply module readiness is missing %s **Cause of error** - While the drive was still under servo control, both the Ready signal of the inverter and the Ready signal of the supply module were switched to inactive. - Possible causes: - UV was switched off over X70 - Fault clearance of the UV: - Dropout in power supply - DC-link voltage is too high - DC-link voltage is too low - DC-link current is too high - PLC or external wiring switched UV off - Noise signals on wiring CC -> UV, CC -> UM - CC controller unit is defective. **Error correction** - Check the diagnostic LEDs on the UV in the event of an error - Check the power supply of the UV - Check the enabling signal on X70 - Check whether the braking resistor is connected for a non-regenerative power supply. - Check the cable ground and shield - Exchange the supply module and the power module - Exchange the CC controller unit. - Inform your service agency.

Error number	Description
231-8900	**Error message** 8900 Noml. speed > %5 1/min: Field weakening inactive %1 **Cause of error** - The field weakening is not activated (machine parameter ampVoltProtection = 0) - The nominal speed was higher than the rotational speed possible without field weakening **Error correction** - Activate the field weakening (set machine parameter ampVoltProtection to be unequal to 0)
231-8910	**Error message** 8910 Velocity too high for rotor position measurement %1 **Cause of error** - An excessively high velocity was detected during the rotor position measurement - Cause 1: The rotor position is measured at a vertical axis without counterweight - Cause 2: The motor is in a poor switch-on position. The axis controls a certain position. **Error correction** - Regarding cause 1: Measure the rotor position with another method. Only methods that measure the rotor position at standstill are allowed. - Regarding cause 2: Restart the rotor position measurement - Inform your service agency
231-89F0	**Error message** 89F0 PLC input inactive %s **Cause of error** - A high-speed input is defined in MP4130.0 that stays inactive when the motor is switched on. - The high-speed input is not enabled over W522. **Error correction** - Inform your service agency - Check the PLC program - Check the high-speed PLC input (MP4130.5)

Error number	Description
231-8A00	**Error message** 8A00 No inverter enabling %1 **Cause of error** - Power-on of the drive not possible due to missing enabling of the inverter via –SH1 - Charging contactor and main contactor on the supply unit is not on (e.g. connector X70 on UV) - Safety relay not on (e.g. connectors X71 and X72 of the UV, X73 of the HEIDENHAIN expansion board for Simodrive) - PWM bus cable interrupted - Interruption in the electrical cabinet (unit bus, PWM ribbon cable) - Defective inverter, (supply unit and/or power modules, compact inverter) - Defective PWM interface on the control **Error correction** - Inform your service agency - Check the wiring
231-8A10	**Error message** 8A10 AC fail %1 **Cause of error** - Power-on of the drive not possible, because an AC-fail signal (power supply) is active. - At least one phase is missing at the primary connection of the supply module **Error correction** - Inform your service agency - Check the wiring of the power supply - Test the power supply
231-8A20	**Error message** 8A20 Powerfail %1 **Cause of error** - Power-on of the drive not possible, because a powerfail signal (power supply) is active **Error correction** - Inform your service agency - Check the wiring of the power supply - Test the power supply - DC-link voltage

Error number	Description
231-8A30	**Error message** 8A30 Drive enabling (I32) %1 **Cause of error** - Power-on of the drive not possible due to missing drive enabling via I32 **Error correction** - Inform your service agency - Check the wiring of the emergency-stop loop - Check the wiring for the drive enabling conditions (e.g. door contact, permissive button) - Measure 24 V- at connector X42/pin 33
231-8A40	**Error message** 8A40 Enabling of axis group %1 **Cause of error** - Because of missing drive enabling for axis groups (X150/X151), the drive cannot be switched on. **Error correction** - Inform your service agency - Check the connector on X150/X151 for correct fit - Check the wiring of X150/X151 - Check the optional configuration datum driveOffGroup.
231-8A50	**Error message** 8A50 Inverter not ready %1 **Cause of error** - Power-on of the drive not possible, because an inverter is not ready (RDY signal). - No pulse release, 24 V is missing on the terminal X71 or X72 - On interface PCBs for Siemens inverters, the second axis is not enabled - Compact inverter, inverter supply unit or power module is defective - Interruption at inverter bus cable (supply bus, unit bus, PWM bus) - Defective PWM interface on the control **Error correction** - Correct the defect in the pulse inhibitor on power supply module UV, X71/72 - Replace the defective compact inverter, supply unit or power module - Replace the defective cable - Inform your service agency

Error number	Description
231-8A60	**Error message** 8A60 Field angle incorrect %1 **Cause of error** The drive cannot be switched on because of missing field angle information. **Error correction** - Inform your service agency - Check the entries of the motor table (measuring device) - If necessary, ascertain the field angle
231-8A70	**Error message** 8A70 Drive change active %1 **Cause of error** The drive cannot be switched on because of a head exchange or a wye/delta switchover. **Error correction** - Inform your service agency - Check the PLC program
231-8A80	**Error message** 8A80 Error ack. missing %1 **Cause of error** The drive cannot be switched on because of a missing error acknowledgment. **Error correction** - Press the emergency stop and switch on again - Switch on the control voltage - Inform your service agency - Check the wiring of the emergency stop

Error number	Description
231-8A90	**Error message** 8A90 Safety module %1 **Cause of error** - Not a safety-oriented control: - The drive cannot be switched on because the safety module is locked - Switch-off of the drive through removal of the external drive enabling at the input of the safety module - Safety module is defective (exchange) - Safety-oriented control: - Drive cannot be switched on because of the operating status of the machine - Drive was switched off because of a change to an illegal operating status of the machine **Error correction** - Inform your service agency - Safety module is connected only to CC: - Switch the external drive enabling over the corresponding input at the safety module (apply 24 V) - Safety modules are connected at MC and CC: - All drives: - Check the function of the protective door contact - Cancel the emergency stop - Only spindle drive: - Check the tool holder (closing) - Check the permissive key - Check the position of the detachable-key switch - Exchange the safety module(s)
231-8AA0	**Error message** 8AA0 Illegal reference run %1 **Cause of error** - A reference value was requested during an active touch probe cycle - Internal software error **Error correction** - Inform your service agency. - Check the software version.
231-8AB0	**Error message** 8AB0 Illegal probing %1 **Cause of error** - A touch probe cycle was started during an active reference run - Internal software error **Error correction** - Inform your service agency. - Check the software version.

Error number	Description
231-8AD0	**Error message** 8AD0 Drive enabling gone (Signal: "-STO.A.MC.WD" active) **Cause of error** - It is not possible to switch the drives on because there is no drive release - The MC does not trigger the switch-off signal '-STO.A.M-C.WD' - Hardware defective - Internal software error **Error correction** - Inform your service agency
231-8AE0	**Error message** 8AE0 Drive enabling missing in axis %1 (signal: %4) **Cause of error** - The switch-on procedure was canceled due to an existing error condition: 1 = Signal -ES.A (emergency stop MC) is active during switch-on (emergency stop might have been pressed during switch-on) 2 = Signal -ES.A.HW (emergency stop MC, handwheel) is active during switch-on (emergency stop might have been pressed during switch-on) 3 = Signal -ES.B (emergency stop CC) is active during switch-on (emergency stop might have been pressed during switch-on) 4 = Signal -ES.B.HW (emergency stop CC, handwheel) is active during switch-on (emergency stop might have been pressed during switch-on) 5 = Switch-on procedure not allowed because of a unacknowledged error 6 = Internal software error: addressed axis module/gate array does not exist 7 = Internal error signal -STO.B.CC.WD active during switch-on 8 = Internal error signal -N0 active during switch-on 9 = Internal error signal PWM error active during switch-on **Error correction** - Check the emergency-stop wiring - Inform your service agency

Error number	Description
231-8AF0	**Error message** 8AF0 Encoder %1 defective **Cause of error** - Scale or scale tape contaminated or defective - Scanning head contaminated or defective - Signal cable defective - Encoder input defective on the control - Penetration of humidity - The position encoder is contaminated - Motor encoder cable defective - Motor control board defective **Error correction** - Inform your service agency - Exchange the position encoder - Check the encoder cable - Exchange the motor control board (or better, the control)
231-8B10	**Error message** 8B10 Wrong traverse direction %1 **Cause of error** - DIR entry in motor table is incorrect. - Incorrect power connection of the motor. **Error correction** - Inform your service agency. - Check the DIR entry in the motor table. - Check the power connection of the motor.
231-8B20	**Error message** 8B20 Error %1 field orientation **Cause of error** - Field orientation impossible for mechanical reasons - Incorrect relation between electrical field and mechanical motor movement - Incorrect motor encoder signal - Error in motor connection - Mechanical brake not released **Error correction** - Inform your service agency - Check the configuration data for number of signal periods and distance for the number of signal periods. - Check the configuration datum for the linear distance of one motor revolution - For linear motors: check column STR of the motor table - Check the speed encoder connection - Check the motor connection - Release brakes during orientation

Error number	Description
231-8B30	**Error message** 8B30 Motor temp. %1 too high **Cause of error** - Measured motor temperature is too high - No temperature sensor - Motor encoder cable is defective (wire broken) - Entry in motor table is incorrect - Incorrect or defective temperature sensor was installed - Signal connector: Poor contact **Error correction** - Let the motor cool down - Inform your service agency - Check the motor encoder cable - Check the entry in the motor table - Measure the temperature sensor
231-8B40	**Error message** 8B40 No drive release %1 **Cause of error** - Inverter not ready - No pulse release for the power module - Uz too high - Power-fail signal active - With M controls: I32 input inactive - With P controls: Drive enabling at X50 inactive In addition, for 246 261-xx (digital current controller): - For the given axis an illegal motor model (e.g. linear motor) was selected. - The CC receives a "Drive on" command for a nonexistent axis. - The power module is not ready when the field orientation starts. Readiness of the power module is detected through the Ready signal on the PWM cable. - The power module is not ready when the current controller adjustment begins. **Error correction** - Inform your service agency - Check the activation and wiring of the pulse release - Check Uz - Check the emergency stop circuit - For a non-regenerative system: Is the braking resistor connected? - For a regenerative system: Is the energy recovery activated? - Check the cable ground and shield - Exchange the power module - For SIEMENS power converter (inverter): Exchange the interface card - Exchange the drive control board

Error number	Description
231-8B50	**Error message** 8B50 Axis module %1 not ready **Cause of error** - Safety relay not on (e.g. connectors X71 and X72 of the UV, X73 of the HEIDENHAIN expansion board for Simodrive) - PWM bus cable interrupted - PWM interface on the control defective - Defective axis module - No pulse release for the power module **Error correction** - Inform your service agency - Exchange the power module - Exchange the HEIDENHAIN expansion board for Simodrive - Exchange the PWM bus cable
231-8B60	**Error message** 8B60 Overcurrent cutoff %1 **Cause of error** - Undervoltage, temperature or short-circuit monitor of an IGBT in the inverter has responded. **Error correction** - Let the inverter cool down - Inform your service agency - Check the current controller - Check the motor connection for a short circuit - Check the motor for a short circuit in the windings - Exchange the power module
231-8B70	**Error message** 8B70 External drive lock %1 **Cause of error** - Drive switch-on is blocked by one or more external signals. **Error correction** - Inform your service agency. - Check the external enabling signal (EMERGENCY STOP, PFAIL, N0). - Check the PLC program. - Check the external wiring.
231-8B80	**Error message** 8B80 External drive stop %1 **Cause of error** - The drive is switched off by an external signal. **Error correction** - Inform your service agency. - Check the external enabling signal (EMERGENCY STOP, PFAIL, N0). - Check the PLC program. - Check the external wiring.

Error number	Description
231-8B90	**Error message** 8B90 Current controller not ready %1 **Cause of error** - Error code: %x - The power module or current controller is not ready after switch-on. - The drive is not at a standstill when switched on. **Error correction** - Inform your service agency - Check the pulse inhibitor - Check the rotational speed input
231-8BA0	**Error message** 8BA0 Incorrect reference signal or line count %1 **Cause of error** - Incorrect entry for the line count under STR in the motor table - Faulty reference signal - Noise signals - Motor encoder cable defective (break or short circuit) - Scale or scale tape contaminated or defective - Scanning head contaminated or defective - Signal cable defective - Encoder input defective on the control - Penetration of humidity **Error correction** - Inform your service agency - Check the entry in the motor table - Check the signals of the speed encoder (PWM 8) - Check the encoder cable for breaks or short circuits under mechanical load (folding, stretching, etc.) - Check the shield and its connection in the encoder cable - Exchange the encoder cable - Exchange the motor
231-8BB0	**Error message** 8BB0 Motor temperature too low %1 **Cause of error** - Measured motor temperature too low - Temperatur sensor incorrectly wired (short circuit) - Temperature sensor is defective - Incorrect temperature sensor (KTY84 required) - Hardware error on encoder input board **Error correction** - Inform your service agency - Check the wiring - Test the temperature sensor - Deselect the monitoring for low temperature limit CfgServo-Motor->MotEncCheckOff bit 5 - Exchange the encoder input board

Error number	Description
231-8BC0	**Error message** 8BC0 Motor current %1 too high **Cause of error** - Incorrect motor or power module selected. - Volts-per-hertz spindle: Ramp gradient too high - Power module defective - Motor cable defective (short circuit) - Motor defective (short circuit, ground fault) - Humidity has entered the motor **Error correction** - Inform your service agency - Are the correct motor and power module selected? - Volts-per-hertz spindle: Readjust the ramp gradient (OEM) - Check the motor and motor cable for a ground circuit or short circuit - Exchange the power module
231-8BD0	**Error message** 8BD0 Excessive servo lag in %1 **Cause of error** - The following error of a moving axis is greater than the value specified in the configuration datum CfgPosControl->servoLagMax1. - Overloaded drive - Insufficient lubrication - Mechanical stiffness - Machine vibration - Hardware error in the control loop - For analog axes, servo defective **Error correction** - Reduce the contouring feed rate, increase the rotational speed - Check the lubrication - Remove mechanical stiffness - Remove any possible sources of vibration - Check the parameter CfgFeedLimits->maxAcceleration - Analog axes: Check the servo - Inform your service agency if the error occurs frequently
231-8BE0	**Error message** 8BE0 Encoder defective %1 **Cause of error** - Incorrect nominal increment between two reference marks **Error correction** - Inform your service agency. - Check the entry in the motor table or CfgAxisHardware->posEncoderRefDist. - Check whether the reference signal is disturbed.

Error number	Description
231-8BF0	**Error message** 8BF0 Velocity of the spindle is too low %1 **Cause of error** - Traversing speed of the spindle is to slow because of overload - If the milling power is too high (high power consumption), it lowers the rotational speed of the spindle **Error correction** - Reduce the plunging depth or traversing speed - Inform your service agency
231-8C00	**Error message** 8C00 Encoder on speed input %1 is defective **Cause of error** - There is no motor encoder signal - There is a break in the motor encoder cable - The signal amplitude of the motor encoder is missing or too small - The motor encoder is contaminated - Incorrect parameters of posEncoderResistor during use of an external Y cable **Error correction** - Check the motor encoder connection - Check the motor encoder for proper function - Check the amplitude of the motor encoder signal - Check configuration in machine parameter PosEncoderResistor. If an external Y cable is used, a 1 has to be entered in this machine parameter. - Inform your service agency
231-8C10	**Error message** 8C10 Encoder on speed input %1 is defective (EnDat) **Cause of error** - Position value of the motor encoder is invalid - Interruption in motor encoder cable - Motor encoder defective **Error correction** - Check the motor encoder connection - Check the motor encoder - Inform your service agency

Error number	Description
231-8C20	**Error message** 8C20 Position encoder %1 defective **Cause of error** - There is no position encoder signal - Interruption of the position encoder cable - The signal amplitude of the position encoder is missing or too small - The position encoder is contaminated - Incorrect values of the machine parameter posEncoderResistor during use of an external Y cable - Incorrect values of the machine parameter posEncoderFreq at feed rates greater than 50 kHz (counting frequency results from the encoder resolution) **Error correction** - Check the position encoder connection - Check the function of the position encoder - Check the amplitude of the position encoder. - Check settings in machine parameter PosEncoderResistor. If an external Y cable is used, a 1 has to be entered in this machine parameter. - Check the setting of the machine parameter posEncoderFreq and max. feed velocity. At feed rates greater than 50 kHz (corresponding counting frequency at the position input results from the resolution of the encoder), a 1 must be entered in posEncoderFreq. - Inform your service agency
231-8C30	**Error message** 8C30 Position encoder %1 defective (EnDat) **Cause of error** - Position value of the position encoder is invalid - Interruption in position encoder cable - Position encoder is defective **Error correction** - Check the position encoder connection - Check the position encoder - Inform your service agency
231-8C40	**Error message** 8C40 Speed input %1 measured value not saved (EnDat) **Cause of error** - Position value of the motor encoder was not latched - Interruption in motor encoder cable - Motor encoder defective **Error correction** - Check the motor encoder connection - Check the motor encoder - Inform your service agency

Error number	Description
231-8C50	**Error message** 8C50 Position input %1 measured value not saved (EnDat) **Cause of error** - Position value of the motor encoder was not stored - Interruption in position encoder cable - Position encoder is defective **Error correction** - Check the position encoder connection - Check the position encoder - Inform your service agency
231-8C60	**Error message** 8C60 Signal frequency at encoder of speed input %1 **Cause of error** - Noise on speed encoder signals **Error correction** - Check the encoder signals - Check the shielding - Inform your service agency
231-8C70	**Error message** 8C70 Signal frequency at position encoder %1 **Cause of error** - Noise on position encoder signals **Error correction** - Check the encoder signals - Check the shielding - Inform your service agency
231-8C80	**Error message** 8C80 Amplitude too high at encoder on speed input %1 **Cause of error** - The amplitude of the encoder signal is too high or the signal for contamination is active. - Noise on motor encoder signal - Short circuit in the motor encoder cable - Signal amplitude of motor too high **Error correction** - Check the motor encoder connection (ground connection) - Check the motor encoder - Inform your service agency

Error number	Description
231-8C90	**Error message** 8C90 Amplitude too high at position encoder %1 **Cause of error** - The amplitude of the position encoder signal is too high or the signal for contamination is active - Noise on the encoder signal - Short circuit in the encoder cable - Encoder signal amplitude too high - Incorrect values in the machine parameter posEncoderResistor **Error correction** - Check the encoder connection (ground connection) - Check the encoder - Check configuration in machine parameter posEncoderResistor. If a position encoder is used (no external Y cable), a 0 has to be entered in this machine parameter. - Inform your service agency
231-8CA0	**Error message** 8CA0 Incorrect reference signal or line count %1 **Cause of error** - Incorrect entry for the line count under STR in the motor table - Faulty reference signal - Noise signals - Motor encoder cable defective (break or short circuit) **Error correction** - Check the entry in the motor table - Check the signals of the speed encoder (PWM 8) - Check the encoder cable for breaks or short circuits under mechanical load (folding, stretching, etc.) - Check the shield and its connection in the encoder cable - Exchange the encoder cable - Exchange the motor - Inform your service agency
231-8CB0	**Error message** 8CB0 Commutation angle %1 missing **Cause of error** The commutation angle necessary for operating the motor is missing. **Error correction** - Determine the commutation angle again - In the machine parameters, change the procedure for determining the commutation angle - Inform your service agency

Error number	Description
231-9200	**Error message** 9200 Parameter compIpcJerkFact is faulty **Cause of error** - On the CC, the input range for the "compIpcJerkFact" parameter is 0.0 to 0.8 **Error correction** - Check the input value in machine parameter "compIpcJerk-Fact" - Inform your service agency
231-9210	**Error message** 9210 Parameter vCtrlDiffGain %1 is too large **Cause of error** - Differential factor is too large (max. value 0.5 [As^2/rev]) **Error correction** - Enter a value less than 0.5 in machine parameter "vCtrlDiff-Gain" - Inform your service agency
231-9220	**Error message** 9220 Incorrect speed encoder input **Cause of error** - Incorrect entry in the "speedEncoderInput" machine parameter - The assignment of the speed encoder input to the PWM output is incorrect **Error correction** - Check the "speedEncoderInput" machine parameter - Inform your service agency
231-9230	**Error message** 9230 Unknown motor type **Cause of error** - Incorrect motor type in the motor table - Unsupported motor type in the motor table - Incorrect motor data in the motor table **Error correction** - Inform your service agency

Error number	Description
231-9240	**Error message** 9240 Incorrect encoder type %1 **Cause of error** - Selected encoder does not match the connected one Examples: EnDat is selected, but an incremental encoder is connected EnDat 2.1 is selected, but EnDat 2.2 is connected - The selected encoder is not supported by this CC **Error correction** - Check the machine parameter "motEncType" (or SYS in the motor table) - Inform your service agency
231-9250	**Error message** 9250 Motor encoder: EnDat 2.2 not possible %1 **Cause of error** - EnDat communication error - An encoder with EnDat 2.2 interface is selected in the motor table, although no EnDat 2.2 encoder is connected - The EnDat 2.2 protocol cannot be read **Error correction** - Check whether the encoder supports EnDat 2.2 - Check the motor table (SYS column) - Check the machine parameter "motEncType" - Check the cable ground and shield - Exchange the motor control board - Check the cable (compare the cable ID number with the documentation) - Check the speed encoder cable (defective or too long) - Check the speed encoder - Inform your service agency
231-9260	**Error message** 9260 Motor parameters changed **Cause of error** - The motor type was changed without switching off the drive **Error correction** - Inform your service agency

Error number	Description
231-9261	**Error message** 9261 CC%2: algebraic signs not consistent in parameters %1 **Cause of error** The entry DIR in the motor table or machine parameter signCorrActualVal is set incorrectly **Error correction** Correct the entry DIR in the motor table or machine parameter signCorrActualVal. The following rule applies: DIR and signCorrActualVal must be set to the same value. Refer to the additional information in the Technical Manual, in the chapter "Defining the traverse direction".
231-9270	**Error message** 9270 Rated speed of motor unknown **Cause of error** - The entry in the motor table is zero **Error correction** - Inform your service agency
231-9280	**Error message** 9280 Parameter filter 1 invalid %1 **Cause of error** - Incorrect entry in machine parameter "vCtrlFiltDamping1," "vCtrlFiltFreq1," "vCtrlFiltType1" or "vCtrlFiltBandWith1" - Internal software error **Error correction** - Check the entry in machine parameter "vCtrlFiltDamping1," "vCtrlFiltFreq1," "vCtrlFiltType1" or "vCtrlFiltBandWith1" - Check the software version - Inform your service agency
231-9290	**Error message** 9290 Parameter filter 2 invalid %1 **Cause of error** - Incorrect entry in machine parameter "vCtrlFiltDamping2," "vCtrlFiltFreq2," "vCtrlFiltType2" or "vCtrlFiltBandWith2" - Internal software error **Error correction** - Check the entry in machine parameter "vCtrlFiltDamping2," "vCtrlFiltFreq2," "vCtrlFiltType2" or "vCtrlFiltBandWith2" - Check the software version - Inform your service agency

Error number	Description
231-92A0	**Error message** 92A0 Parameter filter 3 invalid %1 **Cause of error** - Incorrect entry in machine parameter "vCtrlFiltDamping3," "vCtrlFiltFreq3," "vCtrlFiltType3" or "vCtrlFiltBandWith3" - Internal software error **Error correction** - Check the entry in machine parameter "vCtrlFiltDamping3," "vCtrlFiltFreq3," "vCtrlFiltType3" or "vCtrlFiltBandWith3" - Check the software version - Inform your service agency
231-92B0	**Error message** 92B0 Parameter filter 4 invalid %1 **Cause of error** - Incorrect entry in machine parameter "vCtrlFiltDamping4," "vCtrlFiltFreq4," "vCtrlFiltType4" or "vCtrlFiltBandWith4" - Internal software error **Error correction** - Check the entry in machine parameter "vCtrlFiltDamping4," "vCtrlFiltFreq4," "vCtrlFiltType4" or "vCtrlFiltBandWith4" - Check the software version - Inform your service agency
231-92C0	**Error message** 92C0 Parameter filter 5 invalid %1 **Cause of error** - Incorrect entry in machine parameter "vCtrlFiltDamping5," "vCtrlFiltFreq5," "vCtrlFiltType5" or "vCtrlFiltBandWith5" - Internal software error **Error correction** - Check the entry in machine parameter "vCtrlFiltDamping5," "vCtrlFiltFreq5," "vCtrlFiltType5" or "vCtrlFiltBandWith5" - Check the software version - Inform your service agency
231-92D0	**Error message** 92D0 Line count of motor has changed **Cause of error** - Line count of the motor has changed, although it was not switched off. **Error correction** - Inform your service agency

Error number	Description
231-92E0	**Error message** 92E0 Line count for motor encoder incorrect %1 **Cause of error** - Encoder line count in the parameter is not equal to the EnDat line count found - The entry in machine parameter "cfgServoMotor --> motStr" is incorrect - Incorrect entry under STR in motor table - EnDat 2.2 motor encoder without line count information must be defined as STR = 1 or "cfgServoMotor --> motStr=1" **Error correction** - Edit the machine parameter "cfgServoMotor --> motStr" - Change STR in motor table - Inform your service agency

Error number	Description
231-92F0	**Error message** 92F0 Incorrect axis assignment **Cause of error** - Incorrect axis assignment with moment master-slave operation - Master and Slave are not assigned to the same controller motherboard - Axes in master-slave torque control are permitted only at the following pairs of encoder inputs: 6-axis CC424 X15 and X17 X16 and X18 6-axis CC 6106 X15 to X20 8-axis CC424: X15 and X17 X16 and X18 X19 and X80 X20 and X81 8-axis CC 6108 X15A to X18A X15B to X18B 10 axis CC424: X15 and X17 X16 and X18 10-axis CC 6110 X15A to X18A X15B to X20B 12-axis CC424: X15 and X17 X16 and X18 X82 and X84 X83 and X85 14-axis CC424: X15 and X17 X16 and X18 X19 and X80 X20 and X81 X82 and X84 X83 and X85 16-axis CC424: X15 and X17 X16 and X18 X19 and X80 X20 and X81 X82 and X84 X83 and X85 X86 and X88 X87 and X89 **Error correction** - Change the axis assignment - Inform your service agency

Error number	Description
231-9300	**Error message** 9300 Field angle determination %1 not allowed **Cause of error** - Field angle determination is not allowed in this mode - The method selected for determining the field angle is invalid or not possible with this encoder **Error correction** - Check the entry in machine parameters "motTypeOfFieldAdjust" and "motFieldAdjustMove" and/or the entry in the SYS column of the motor table - Inform your service agency
231-9310	**Error message** 9310 Wrong input for position encoder **Cause of error** - An incorrect input was selected for the position encoder (machine parameter "posEncoderInput") - Possible configuration CC61xx: PWM output <-> position encoder X51 <-> X201 X52 <-> X202 X53 <-> X203 X54 <-> X204 X55 <-> X205 X56 <-> X206 **Error correction** - Check the machine parameter "posEncoderInput" - Inform your service agency
231-9320	**Error message** 9320 Position encoder: EnDat 2.2 not possible %1 **Cause of error** - EnDat communication is defective - An encoder with EnDat 2.2 interface is selected in machine parameter posEncoderType or motEncTyp, although no EnDat 2.2 encoder is connected. - The EnDat 2.2 protocol cannot be read **Error correction** - Check whether the position encoder supports EnDat 2.2 - Check the machine parameter posEncoderType or motEncTyp - Check the cable ground and shield - Exchange the motor control board - Check the cable (compare the cable ID number with the documentation) - Check the position encoder cable (defective or too long) - Check the position encoder - Inform your service agency

Error number	Description
231-9330	**Error message** 9330 Incorrect position encoder input configured %1 **Cause of error** - The configured position encoder input does not exist **Error correction** - Check the configuration of the position encoder inputs - Inform your service agency
231-9340	**Error message** 9340 PWM frequency error %1 **Cause of error** - Entered PWM frequency in parameter "ampPwmFreq" lies outside the permissible input range - PWM frequencies that may not be combined with each other were selected **Error correction** - Check the parameter "ampPwmFreq" - Inform your service agency
231-9350	**Error message** 9350 PWM frequency too high %1 **Cause of error** - PWM frequency selected over 5000 kHz for PWM output X51 or X52, although PWM output X53 or X54 is active **Error correction** - Check machine parameter ampPwmFreq for PWM output X51 or X52 - Deactivate PWM output X53 and/or X54 - Inform your service agency
231-9360	**Error message** 9360 Double speed not possible %1 **Cause of error** - Control loop on X51 or X52 is defined as 'double speed', although the control loop on X53 or X54 is active - Control loop on X55 or X56 is defined as 'double speed', although the control loop on X57 or X58 is active (only CC 4xx with 8 control loops) **Error correction** - Define the control loop on X51 or X52 as 'single speed', or deactivate the PWM output X53 or X54 - Define the control loop on X55 or X56 is defined as 'single speed', or deactivate the PWM output X57 or X58 (only CC 4xx with 8 control loops) - Inform your service agency

Error number	Description
231-9370	**Error message** 9370 "Inverter.inv" is not supported %1 **Cause of error** - The 'Inverter.inv' file is not supported by this software **Error correction** - Replace the 'Inverter.inv' file with 'Motor.amp' - Inform your service agency
231-9380	**Error message** 9380 Voltage protection module parameter %1 **Cause of error** - Incorrect entry in machine parameter "ampVoltProtection" - Internal software error **Error correction** - Check the entry in machine parameter "ampVoltProtection" - Check the software version - Inform your service agency
231-9390	**Error message** 9390 Increased current controller factors: Wrong motor type %1 **Cause of error** - In machine parameter iCtrlAddInfo, the compensation in the current control loop is activated for a motor type other than a synchronous motor (linear motor, asynchronous motor). **Error correction** - Deactivate the compensation for increased current control factors by entering the value "0" in iCtrlAddInfo.
231-93A0	**Error message** 93A0 PDT1 feedforward current contrlr: Wrong motor type **Cause of error** - In machine parameter "iCtrlDiffFreqFF," PDT1 feedforward control was activated for the current control circuit of a drive with e.g. a linear or asynchronous motor. - PDT1 feedforward control is only possible for the current control circuit of a drive with synchronous motor. **Error correction** - Deactivate the PDT1 feedforward by entering the value 0 in machine parameter "iCtrlDiffFreqFF." - Inform your service agency.

Error number	Description
231-93B0	**Error message** 93B0 PDT1 feedforward current contrlr.: Invalid cutoff freq. **Cause of error** - Illegal maximum cutoff frequency in machine parameter "iCtrlDiffFreqFF." - The following are the maximum permissible cutoff frequencies for the respective PWM frequencies: PWM frequency Maximum cutoff frequency 3333 Hz 800 Hz 4000 Hz 960 Hz 5000 Hz 1200 Hz 6666 Hz 1600 Hz 8000 Hz 1920 Hz 10000 Hz 2400 Hz **Error correction** - Enter the permissible cutoff frequency in machine parameter "iCtrlDiffFreqFF." - Inform your service agency.
231-93C0	**Error message** 93C0 INVERTER.INV is faulty %1 **Cause of error** - Incorrect entry in the power module table: INVERTER.INV - Incorrect entry for I-MAX, U-IMAX or R sensor - R sensor not identical to U-IMAX/I-MAX **Error correction** - Check the entries for I-MAX, U-IMAX and R sensor - Inform your service agency
231-93D0	**Error message** 93D0 Transmission ratio is incorrect %1 **Cause of error** The transmission ratio has incorrect parameters: - Faulty entry for distPerMotorTurn, i.e. distance per motor revolution - Incorrect entry in posEncoderIncr, i.e. signal periods of position encoder - Faulty entry for distance traversed after posEncoderDist, i.e. parameterized signal periods of the position encoder **Error correction** - Inspection of the parameters distPerMotorTurn, posEncoderIncr and posEncoderDist - Inform your service agency

Error number	Description
231-93E0	**Error message** 93E0 PWM frequency greater than 5 kHz requires double speed %1 **Cause of error** - If the PWM frequency selected in the "ampPwmFreq" machine parameter is greater than 5000 Hz, the machine parameter "ctrlPerformance" must be set to Double Speed **Error correction** - Either reduce the PWM frequency in "ampPwmFreq," - or set the "ctrlPerformance" machine parameter to Double Speed - Inform your service agency
231-93F0	**Error message** 93F0 Maximum computer power exceeded **Cause of error** - CC61xx: The desired computing power in the "ctrlPerformance" machine parameter is not possible. The following configurations per controller unit are allowed: Single | Double 6 | 0 4 | 1 2 | 2 0 | 3 "ctrlPerformance" machine parameter = 0 Single Speed "ctrlPerformance" machine parameter = 1 Double Speed **Error correction** - Reduce the number of axes per controller unit - Change the double-speed axes to single-speed axes ("ctrlPerformance" machine parameter from 1 to 0) - Inform your service agency
231-9400	**Error message** 9400 Encoders with 11 µA are not supported **Cause of error** - "posEncoderSignal" machine parameter is set to 11 µA **Error correction** - "posEncoderSignal" machine parameter is set to 11 Vpp - Inform your service agency

Error number	Description
231-9410	**Error message** 9410 Relation of posEncoderDist to posEncoderIncr is faulty %1 **Cause of error** - The relation of CfgAxisHardware->posEncoderDist to CfgAxisHardware->posEncoderIncr does not agree with the values from the EnDat encoder - With EnDat 2.2: See the Technical Manual of the control **Error correction** - Check the entries CfgAxisHardware->posEncoderDist or CfgAxisHardware->posEncoderInc - Inform your service agency
231-9420	**Error message** 9420 Configuration of X150 not possible %1 **Cause of error** - The entry in GenDriveOffGroup (axis group switch-off with X150) is invalid **Error correction** - Correct the entry in GenDriveOffGroup - Inform your service agency
231-9430	**Error message** 9430 Configuration of I32 not possible %1 **Cause of error** - The entry in GenEmergencyStopFunction for drive enabling through I32 input is invalid **Error correction** - Correct the entry in GenEmergencyStopFunction - Inform your service agency
231-9440	**Error message** 9440 Configuration of powerfail is not possible %1 **Cause of error** - The entry in AmpAcFailSelection for AC fail or power fail is invalid **Error correction** - Correct the entry in AmpAcFailSelection - Inform your service agency
231-9450	**Error message** 9450 Configuration of PWM pattern not possible %1 **Cause of error** - The entry in ICtrlPwmInfo (configuration of the PWM pattern) is invalid **Error correction** - Correct the entry in ICtrlPwmInfo - Inform your service agency

Error number	Description
231-9460	**Error message** 9460 Configuration of LIFTOFF is not possible %1 **Cause of error** - The entry in PowSupplyLimitOfDcVoltage is invalid - The entry in PowSupplyDcLinkVoltageForSpindleStop is invalid **Error correction** - Correct the entry in PowSupplyLimitOfDcVoltage - Correct the entry in PowSupplyDcLinkVoltageForSpindleS-top - Inform your service agency
231-9470	**Error message** 9470 Configuration of brake output is not possible %1 **Cause of error** - The entry in MotBrakeNotExist is invalid **Error correction** - Correct the entry in MotBrakeNotExist - Inform your service agency
231-9480	**Error message** 9480 Configuration of jerk feedforward is not possible %1 **Cause of error** - The jerk feedforward over ComplpcJerkFact is not possible **Error correction** - Correct the entry in ComplpcJerkFact - Recommendation: As an alternative, the torsion feedfor-ward can be used through CompTorsionFact. - Inform your service agency
231-9490	**Error message** 9490 Configuration of active damping is not possible %1 **Cause of error** - The active damping through CompActiveDampFactor and CompActiveDampTimeConst is not possible **Error correction** - Correct the entry in CompActiveDampFactor and CompActiveDampTimeConst - Inform your service agency

Error number	Description
231-94A0	**Error message** 94A0 Configuration of SyncAxisTorqueDistrFact impossible %1 **Cause of error** - The variable torque distribution through SyncAxis-TorqueDistrFact is not possible **Error correction** - Correct the entry in SyncAxisTorqueDistrFact - Inform your service agency
231-94B0	**Error message** 94B0 Configuration of speed compensation impossible %1 **Cause of error** - The variable shaft speed compensation through SyncAx-isSpeedCorrectRatio is not possible **Error correction** - Correct the entry in SyncAxisSpeedCorrectRatio - Inform your service agency
231-94C0	**Error message** 94C0 DQ PWM freq. > 4 kHz requires double speed %1 **Cause of error** - If the PWM frequency selected in the "ampPwmFreq" machine parameter is greater than 4000 Hz, the machine parameter "ctrlPerformance" must be set to Double Speed **Error correction** - Either reduce the PWM frequency in "ampPwmFreq," - or set the "ctrlPerformance" machine parameter to Double Speed (software option) - Inform your service agency
231-94D0	**Error message** 94D0 DQ-ALM: Check the parameters %1 **Cause of error** - Incorrect supply module selected in CfgPowSupply. - The supply module is not entered in the supply module table Supply.Spy. **Error correction** - Select the correct supply module in CfgPowSupply. - Inform your service agency

Error number	Description
231-94E0	**Error message** 94E0 Switching EnDat 2.2 to 1 Vpp requires reinitialization %1 **Cause of error** - The encoder was changed from EnDat 2.2 to 1 Vpp or vice versa. A reinitialization of the drive is required. **Error correction** - Reinitialize the system - Deselect the axis with the machine parameter axisMode (bit x = 0) - Exit the MP editor - Reactivate the axis in machine parameter axisMode (bit x = 1) and set posEncoderType to the desired value - Re-exit the MP editor - or restart the system
231-94F0	**Error message** 94F0 Impermissible torsion compensation %1 **Cause of error** - Torsion compensation is configured, but the system is a single-encoder system. - Torsion compensation is configured, but stick-slip friction compensation is not configured. **Error correction** - Deactivate torsion compensation with machine parameter compTorsionFact. - Define the assignment between the position encoder inputs and the axes in posEncoderInput (if position encoder is present), and enter the friction compensation in machine parameter compFrictionT2. - Inform your service agency
231-9500	**Error message** 9500 DQ: Inverter not found %1 **Cause of error** - Not able to establish communication with the inverter of the displayed axis. - The DRIVE-CLiQ line is not connected, or incorrectly connected. - The supply voltage of the inverter has been interrupted. - The inverter is defective. - Machine parameter pwmSignalOutput is incorrect. **Error correction** - Check the cabling. - Check the supply voltage of the inverter. - Check the entry in machine parameter pwmSignalOutput. - Inform your service agency

Error number	Description
231-9510	**Error message** 9510 PWM frequency change during active servo control %1 **Cause of error** - The PWM frequency in machine parameter ampPwmFreq must not be changed while the drive or a corresponding drive is still in a control loop. Corresponding axis: X51 - X52 X53 - X54 X55 - X56 X57 - X58 X80 - X81 X82 - X83 X84 - X85 **Error correction** - Deactivate the drive controller before changing machine parameter ampPwmFreq - Inform your service agency
231-9520	**Error message** 9520 Erroneous input in vCtrlTimeSwitchOff %1 **Cause of error** - In machine parameter vCtrlTimeSwitchOff a time of zero was configured. The time entered must permit a safe braking process. - In order to prevent an unbraked runout of axes/spindles without mechanical braking, the time in machine parameter vCtrlTimeSwitchOff must be greater than the maximum possible braking time of the axes that can occur through electrical braking. **Error correction** - Enter an appropriate value in machine parameter vCtrlTimeSwitchOff - Inform your service agency
231-9530	**Error message** 9530 DRIVE-CLiQ axis %1 is still active **Cause of error** When an axis is deactivated in a DRIVE-CLiQ system, the associated axis must likewise be deactivated at the same port (e.g. 301). **Error correction** - Deactivate the second axis that is connected to the same connector (e.g. X301). - The axis must be deactivated through machine parameter axisMode or CfgPlcSStrobe. - Inform your service agency.

Error number	Description
231-9550	**Error message** 9550 EnDat2.2 - FS single-encoder system incorrectly configured %1 **Cause of error** It was configured so that both only the EnDat 2.2 position encoder and only the EnDat 2.2 shaft speed encoder are to be used for functional safety. This is not possible. **Error correction** - Check the entry in Machine Parameter CfgAxisSafety-encoderForSafety: It must either be "speedAndPosEncoder", "speedEncoder" or "posEncoder". - Special settings for single-encoder systems with EnDat 2.2 FS encoders: Set "posEncoder" if only the position encoder is to be used for functional safety or set "speedEncoder" if only the shaft speed encoder is to be used for functional safety - Standard setting: Set "speedAndPosEncoder" if you want to configure a normal dual-encoder system or if the affected axis is run in functional safety with only one encoder (single-encoder safety).
231-9560	**Error message** 9560 MP entry erroneous: counting pulse per path %1 **Cause of error** - Value in machine parameter posEncoderIncr lies outside the permissible range **Error correction** - Check the value in MP_posEncoderIncr and correct it if necessary - Inform your service agency
231-9570	**Error message** 9570 Incorrect parameter entry: path per motor revolution %1 **Cause of error** - Value in machine parameter distPerMotorTurn lies outside the permissible range **Error correction** - Check the value in MP_distPerMotorTurn and correct it if necessary - Inform your service agency

Error number	Description
231-9580	**Error message** 9580 Parameter filter 6 invalid %1 **Cause of error** - Incorrect entry in machine parameter "vCtrlFiltDamping6", "vCtrlFiltFreq6", "vCtrlFiltType6" or "vCtrlBandWidth6" **Error correction** - Check the entry in machine parameter "vCtrlFiltDamping6", "vCtrlFiltFreq6", "vCtrlFiltType6" or "vCtrlBandWidth6" - Inform your service agency
231-9590	**Error message** 9590 Maximum computing power exceeded **Cause of error** -CC61xx: There is not enough computing power for the activated expanded controller functions. **Error correction** - Deactivate the expanded controller functions - Reduce the number of axes per controller unit - Change double-speed axes to single-speed axes (change the machine parameter ctrlPerformance from 1 to 0) - Inform your service agency
231-9591	**Error message** 9591 TRC is not active **Cause of error** TRC is not active because not all parameters have been entered **Error correction** The following parameters are necessary in order to activate TRC: - Mass moment of inertia of the motor (from the motor table) or acceleration feedforward control CfgController-Comp.compAcc - Proportional component of the speed controller CfgSpeed-Control.vCtrlPropGain - Transformation ratio CfgAxisHardware.distPerMotorTurn
231-9800	**Error message** 9800 CC%2 MC command unknown %1 **Cause of error** - MC command is not allowed for this hardware - MC command is not allowed at this time - 0 = error in command code > 255 1...255 = incorrect or invalid command code - Internal software error **Error correction** - Inform your service agency. - Check the software version.

Error number	Description
231-9900	**Error message** 9900 CC%2 CC command unknown %1 **Cause of error** - CC command is not allowed for this hardware - Internal software error **Error correction** - Inform your service agency - Check the software version
231-A001	**Error message** A001 Cancelation of brake-test call monitoring **Cause of error** - Protective door(s) were opened during the brake test - No drive-ready signal during the brake-test call monitoring **Error correction** - Leave the protective door(s) closed during the brake test - Ensure the drive readiness during the brake test
231-A002	**Error message** A002 Cancelation of brake-line-test call monitoring A021 Cancelation of brake line test. Guard door open **Cause of error** - Guard door(s) were open during the brake control test or brake line test **Error correction** - Leave the guard door(s) closed during the brake control test or brake line test
231-A003	**Error message** A003 SPLC commissioning mode active **Cause of error** - CRC check of the SPLC program is deactivated (machine parameter CfgSafety --> commissioning is set) **Error correction** - The commissioning mode must be deactivated before a machine is delivered (reset commissioning in the machine parameter CfgSafety) - Inform your service agency

Error number	Description
231-A004	**Error message** A004 Short circuit on 24 V of the T.BRK.B signal A020 Short circuit on 24 V of the T.BRK.B signal **Cause of error** - Error during braking control test - Signal level = +24 V at input: -T.BRK.B of the SPL module although 0 V is expected due to the control. **Error correction** - Check the external wiring of the brakes and the T.BRK signal. - Check the relay for the brake control. - Inform your service agency.
231-A005	**Error message** A005 CC%2 Warning: Controller unit not suited for FS **Cause of error** When checking the safety kernel software SKERN-CC, the control unit detected, that this hardware (CC, UEC, UMC) is not suitable for control systems with functional safety (FS). This controller unit fulfills the safety requirements according to EN 13849 and is not approved for functional safety. **Error correction** - Exchange the affected controller unit (CC, UEC, UMC) - Inform your service agency.
231-A006	**Error message** A006 SPLC-CC%2: Cross comparison failed, output %4 **Cause of error** - Error in the cross comparison of the SPLC-CC outputs. - SPLC program of the CC[x] defines the output as 0 (LOW). However, the value of the output read from the terminal is 1 (HIGH). **Error correction** - Check the wiring of the output. - Check the SPLC program: signals for setting and resetting the SPLC outputs should always stay stable for at least 2 SPLC cycles. - Inform your service agency.

Error number	Description
231-A040	**Error message** A040 CC%2 operating mode not possible **Cause of error** The status of the operating mode switches and the protective doors is not allowed. **Error correction** - Inform your service agency - Check the setting of the operating mode switches - Check the status of the switch for the safety doors - Check the wiring
231-A041	**Error message** A041 CC%2 SOM 4 not possible **Cause of error** - Keylock switch 1 not in automatic mode (BA1) – Keylock switch 1 defective - Wiring error **Error correction** - Set keylock switch 1 to automatic mode (BA1) - Inform your service agency
231-A042	**Error message** A042 CC%2 SOM 4 not released **Cause of error** The BA4 operating mode is selected by key switch but has not been released. **Error correction** - Inform your service agency.
231-A043	**Error message** A043 CC%2 SOM 2 only one axis allowed **Cause of error** The simultaneous movement of more than one axes is not allowed in operating mode BA2 with open safety doors. **Error correction** Wait until all axes are at standstill, and then start only one axis.

Error number	Description
231-A080	**Error message** A080 CC%2 operating state not equal MC **Cause of error** - The automatic, SRG, SBH, and SH operating conditions are compared cyclically between the MC and CC. If the values remain unequal for longer than 500 ms, a Stop 1 is released. **Error correction** - Acknowledge the error message with CE - Switch on the machine - Inform your service agency - Check the software version
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The requested safety function for the axis group does not match. ("pp_AxGrpStateReq[Var.]", Var.= axis group) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The motion request for an axis or for the axis group does not match. ("pp_AxGrpActivate[Var.]", Var.= axis group) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The axis-specific movement release does not match. ("pp_AxFeedEnable[Var.]", Var.= axis index) **Error correction** - Check the SPLC program - Inform your service agency

Error number	Description
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The request of a stop reaction for the axis group does not match. ("pp_AxGrpStopReq[Var.]", Var.= axis group) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The status of the axis-group-specific permissive button does not match. ("pp_AxGrpPB[Var.]", Var.= axis group) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC The reported condition of the chain of safety relays or chain of normally closed contacts does not match. ("pp_GenFB_NCC", Var.= No meaning) **Error correction** - Check the SPLC program - Inform your service agency

Error number	Description
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The requested or reported condition of the machine control voltage does not match. ("pp_GenCVO", Var.= No meaning) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The axis-group-specific permission for drive enable does not match. ("pp_AxGrpPermitDrvOn", Var.= axis group) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The collective status of the machine keys does not match. ("pp_GenMKG", Var.= No meaning) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The status of the signal for the brake-line test does not match. ("pp_GenTBRK", Var.= No meaning) **Error correction** - Check the SPLC program - Inform your service agency

Error number	Description
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The safety-related mode of operation SOM active in the SKERN does not match. ("pp_GenSOM", Var.= No meaning) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The status of the read-back outputs does not match. ("readBackOutputs[Var.]", Var.= index number of the output) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The status "SPLc program active" does not match. ("running", Var.= No meaning) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the safety core software in a cross-comparison between MC and CC. The "Stop" request does not match. ("stopReq", Var.= No meaning) **Error correction** - Check the SPLC program - Inform your service agency

Error number	Description
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the output data of the safety core software in a cross-comparison between MC and CC The ascertained safety status of the axis group does not match. ("sAxGrpState[Var.]", Var.= axis group) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the output data of the safety core software in a cross-comparison between MC and CC The axis-dependent braking control does not match. ("sAxBrkReleaseReq[Var.]", Var.= axis index) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the output data of the safety core software in a cross-comparison between MC and CC The motion status of the axis group does not match. ("sAxGrpInMotion[Var.]", Var.= axis group) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the output data of the safety core software in a cross-comparison between MC and CC The operating readiness for functional safety (FS) does not match. ("sGenSafe", Var. = No meaning) **Error correction** - Check the SPLC program - Inform your service agency

Error number	Description
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the output data of the SKERN in a cross-comparison between the A channel and B channel. The axis-dependent brake control for multiple brakes per axis does not match. ("sMultiBrkOnAxisReleaseReq[Var.]", Var.= axis index) **Error correction** - Check the SPLC program - Inform your service agency
231-A081	**Error message** A081 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the SKERN in a cross-comparison between the A channel and B channel. The value for the feed rate limit does not match. ("pp_AxFeedMax[Var.]", Var.= axis index) **Error correction** - Check the SPLC program - Inform your service agency
231-A082	**Error message** A082 CC%2 NE2 level does not change after 0 during dyn. test **Cause of error** - During the dynamic test of the 2nd emergency stop loop (every 1.5 minutes at the latest), a temporary change to 0 V level is expected at the input (NE2). If a 0 V or 24 V level remains continuously on inside the test window of 100 ms, it causes the error. - The time window for the dynamic test is too small (computing time problems, software errors) **Error correction** - Check the software version - Inform your service agency - Check the wiring - Check the emergency stop button - Exchange the hardware

Error number	Description
231-A082	**Error message** A082 CC%2 cross comparison no.=%4 Var.=%5 MC=%6 CC=%7 **Cause of error** - A difference was found in the input data of the SKERN in a cross-comparison between the A channel and B channel. The value for the readback channels does not match. ("pp_ReadBackOutputs", Var. = Number of the SPLC output) **Error correction** - Check the voltages and wiring of the SPLC output - Check the SPLC program and correct it if necessary - Inform your service agency
231-A083	**Error message** A083 CC%2 S input not equal to 0 in dynamic test **Cause of error** - During the dynamic test of the 2nd emergency stop loop (no later than every 1.5 min), a 0 V level is expected at all safety-related door-contact keylock and switch inputs for a short period. This error appears if a 24 V level continues throughout the test window of 100 ms. - The time window for the dynamic test is too small (computing time problems, software errors) **Error correction** - Check the software version - Inform your service agency - Check the wiring - Check the door contacts and key switches - Exchange the hardware
231-A084	**Error message** A084 S timeout in stop reaction SS2 %4 ObjId=%5 **Cause of error** - The maximum permissible time for a controlled stop (SS2 – braking on the contour) of the axis group (= ObjId) was exceeded. The maximum permissible time is 30 seconds. **Error correction** - Check the SPLC program and PLC program - Inform your service agency.

Error number	Description
231-A085	**Error message** A085 SKERN-CC%2, X%4: Comm.error during EnDat encoder dynamic sampling **Cause of error** A communication error occurred during forced dynamic sampling of the EnDat encoder. Both ignore flags were set at the same time. **Error correction** Inform your service agency.
231-A086	**Error message** A086 SKERN-CC%2: Communications error during dynamic sampling of X%4 **Cause of error** The dynamic sampling of the EnDat22 encoder could not be performed within the prescribed time. **Error correction** Note further error messages. Inform your service agency.
231-A087	**Error message** A087 SKERN-CC%2: illegal Ignore bit for EnDat forced dynamic sampling **Cause of error** An active ignore bit was detected outside of an EnDat forced dynamic sampling. **Error correction** - Restart the control - If the error recurs, inform your service agency
231-A090	**Error message** A090 Drive lock through safety software **Cause of error** - Emergency stop activated (by CC) - A drive is to be switched on, although the system in in the "emergency stop" condition ("-ES.B" or "-NE2" signal is active). - Internal software error **Error correction** - Check the wiring - Check the software version - Inform your service agency

Error number	Description
231-A091	**Error message** A091 Drive lock through safety software **Cause of error** - Emergency stop activated (of MC) - A drive is to be switched on, although the system in in the "emergency stop" condition ("-ES.A" or "-NE1" signal is active). - Internal software error **Error correction** - Check the wiring - Check the software version - Inform your service agency
231-A092	**Error message** A092 Drive lock through safety software **Cause of error** - System test is active - A drive is to be switched on, although the system is still in a test program - Internal software error **Error correction** - Check the software version - Inform your service agency
231-A093	**Error message** A093 Drive lock through FS, stop 1 switch-off active **Cause of error** - Drive locked through functional safety FS - A drive was supposed to be switched on, although the CC has not yet completed a completed a running stop 1 switch-off - Internal software error **Error correction** - Check the software version - Inform your service agency
231-A094	**Error message** A094 Drive lock of spindle, tool holder opened **Cause of error** - A spindle drive was supposed to be switched on while the guard door was open, although the tool holder was open. - Internal software error **Error correction** - Check the wiring - Check the software version - Inform your service agency

Error number	Description
231-A095	**Error message** A095 Spindle drive lock, status of permissive button invalid **Cause of error** - A spindle drive was supposed to be switched on while the guard door was open, although the permissive button is not pressed or was not let go beforehand. - Internal software error **Error correction** - Check the wiring - Check the software version - Inform your service agency
231-A096	**Error message** A096 Spindle drive lock, operating mode is not allowed **Cause of error** - A spindle drive was supposed to be switched on while the guard door was open, although the keylock switch was in the 'BA1' position ('unqualified operator'). - Internal software error **Error correction** - Check the keylock switch position - Check the wiring - Check the software version - Inform your service agency
231-A097	**Error message** A097 Drive lock on spindle, SS2 reaction active **Cause of error** - A spindle drive was supposed to be switched on while the guard door was open, although a stop2 reaction is active for the spindle. - Internal software error **Error correction** - Check the software version - Inform your service agency
231-A098	**Error message** A098 Drive lock on spindle, STO is active **Cause of error** - A spindle drive was supposed to be switched on while the guard door was open, although the STO safety function (safely switched-off torque) is still active. - Internal software error **Error correction** - Check the software version - Inform your service agency

Error number	Description
231-A099	**Error message** A099 CC%2 Drive lock - non-deletable system error **Cause of error** - Drive switch-on was prevented because a non-deletable system error occurred. **Error correction** - Check the cause of the system error (see log entry) and correct if if possible. - Inform your service agency
231-A210	**Error message** A210 Faulty control of brake %1: Step %4 **Cause of error** - Incorrect signal level at the FS input: "-T.BRK.B" during brake control test Test step 2 = Release brake through B channel output: +24 V detected although T.BRK should provide 0 V Test step 3 = Release brake through A channel output: +24 V detected although T.BRK should provide 0 V Test step 4 = Release brake through A and B channel outputs: 0 V detected although T.BRK should provide +24 V **Error correction** - Check external wiring of the motor brake - Generate the service files and notify the service agency
231-A800	**Error message** A800 CC limit switch %1+ **Cause of error** Permissible positive range of traverse exceeded. **Error correction** - Inform your service agency - Check the parameter setting CfgPositionLimits->swLimitSwitchPos1
231-A810	**Error message** A810 CC limit switch %1- **Cause of error** Permissible negative traverse range exceeded. **Error correction** - Customer service agency - Check the setting of parameter CfgPositionLimits->swLimitSwitchNeg1

Error number	Description
231-A820	**Error message** A820 CC speed greater than SRG %1 **Cause of error** The maximum permissible velocity in the SRG operating mode was exceeded. **Error correction** - Reduce the feed rate and shaft speed before opening the safety doors - Check the operating mode (setting of key-operated switch) - Inform your service agency - Check the parameter values
231-A830	**Error message** A830 CC SRG rot. speed = 0 %1 **Cause of error** - The maximum permissible velocity is 0 (as per parameter) - Only spindle drive: the parameter values for the gear ranges are 0 - An illegal operating mode was selected by keylock switch - There is a circuit error, or a disturbance at input I19 of the operator safety module - Internal software error **Error correction** - Check the operating mode version (position of keylock switch) - Check the wiring of the operator safety module inputs. - Check the parameter values - Inform your service agency
231-A840	**Error message** A840 CC SBH rot. speed too high %1 **Cause of error** - The maximal permissible velocity of the standstill monitoring was exceeded. **Error correction** - Inform your service agency - Check the drive
231-A850	**Error message** A850 CC SBH rot. speed = 0 %1 **Cause of error** - Standstill monitoring was set to 0 - Internal software error **Error correction** - Inform your service agency - Check CfgAxisHardware->transmission (traverse per motor revolution)

Error number	Description
231-A860	**Error message** A860 Traverse in SRG too large %1 **Cause of error** - The maximum permissible traverse in the SRG operating mode was exceeded because: - During probing with oriented spindle stop, the spindle axis was moved by more than 2 revolutions. **Error correction** - Check the probing sequence - Close the protective doors - Inform your service agency
231-A870	**Error message** A870 Brake test is not allowed %1 A200 Brake test is not allowed %1 **Cause of error** - Drive ready signal (RDY signal) or inverter enabling signal is missing. - Protective door(s) are not closed, although the parameter settings require it. - The brake to be tested is not assigned to this drive **Error correction** - Check the inverter enabling signals (wiring, PLC program or SPLc program) - Close the protective door(s) - Check the parameter settings, and edit them if necessary. - Inform your service agency
231-A880	**Error message** A880 1st violation of positive software limit range %1 **Cause of error** - The drive moved into the positive limit-switch area for the 1st time **Error correction** - Move the drive out of the positive limit-switch area - Check the parameters for limit-switch areas, and edit them if necessary.
231-A890	**Error message** A890 1st violation of negative software limit range %1 **Cause of error** - The drive moved into the negative limit-switch area for the 1st time **Error correction** - Move the drive out of the negative limit-switch area - Check the parameters for limit-switch areas, and edit them if necessary.

Error number	Description
231-A8A0	**Error message** A8A0 CC nominal-to-actual deviation of position values %1 **Cause of error** - The safety function nominal/actual value monitoring of position values has ascertained an excessive deviation. - The motor is moving while the slide is not, or vice versa - The mechanical motion transmission is interrupted - Thermal expansion of mechanical transmission components - The transmission ratio of the motor to the position encoder is incorrect (machine parameter distPerMotorTurn) - Improper installation of position encoder on ball screw - The entry in the axis-specific parameter maxPosDiff is defined too small **Error correction** - Check the mechanical motion transmission - Check the encoder fastening - Check the thermal expansion of the mechanical transmission components (e.g. recirculating ball screw) - Check the transmission ratio of the motor to the position encoder - Check the installation of the position encoder on the ball screw - Inform your service agency.
231-A8C0	**Error message** A8C0 Drive lock through FS, %1 has not been checked **Cause of error** - Drive locked through functional safety FS - A non-checked drive moved in an operating mode other than "reference run" while the guard door was open. - Internal software error **Error correction** - Check the software version - Inform your service agency
231-A8D0	**Error message** A8D0 SS1 still active—switch-on not allowed %1 **Cause of error** The safety software of the controller unit (SKERN-CC) has prevented the drive from being switched back on. A triggered SS1 stop reaction was not yet fully concluded. **Error correction** - Try to switch the drive on again. - If the problem recurs: Check the process of the SPLC and PLC program and inform your service agency.

Error number	Description
231-A8E0	**Error message** A8E0 Timeout during braking %1 MC timeout during braking (SS2) %2 **Cause of error** - The maximum permissible time for a controlled stop (SS2 – braking on the contour) was exceeded **Error correction** - Check the parameter values: timeLimitStop2: Default time for bringing axes to a controlled standstill for SS2 reaction - Inform your service agency
231-A8F0	**Error message** A8F0 Drive lock %1 - safety function STO is active **Cause of error** - Drive switch-on was prevented because the safety function STO is presently active for this drive. - The STO safety function could not be exited. Exiting the STO safety function requires the following: The activation of the axis group through the SPLC program (PP_AxGrpActivate of the associated axis group) Enabling of the drive by the SPLC program (PP_AxFeedEnable of the associated axis) The switch-on command of the PLC program (PLC Module 9161) - If the message appears during the current controller adjustment of the spindle, the "spindle start" key was not pressed before the adjustment was started. **Error correction** - Press the "spindle start" key before the spindle adjustment - Press the key for moving the axis on the machine control panel for a longer time - Optimize program processes in the PLC and SPLC program - Check the SPLC program and correct if necessary - Inform your service agency
231-A900	**Error message** A900 S traverse range exceeded when braking at contour (SS2) %1 **Cause of error** - When braking at a contour (SS2), the maximum permissible path in the safety-related machine parameter distLimitStop2 was exceeded. **Error correction** - Check the parameter value: distLimitStop2: Axis-specific limit value for maximum permissible path upon SS2 reaction. - Inform your service agency

Error number	Description
231-A910	**Error message** A910 CC Speed/noml. value deviation too large %1 **Cause of error** - The actual velocity deviates for longer than the permissible duration (feed axes: machine parameter timeToleranceSpeed) by the maximum permissible deviation (feed axes: machine parameter speedDiffNom) from the nominal velocity value. - Possible causes: + Machine parameter "timeToleranceSpeed" is defined too small + Machine parameter "speedDiffNom " is defined too small. + I component of the speed controller in the machine parameter "vCtrlIntGain" is defined too small. + Is the speed encoder cable connected? + Motor encoder defective or loose **Error correction** - Check the machine parameter "timeToleranceSpeed" - Check the machine parameter "speedDiffNom" - Check the I component of the speed controller machine parameter "vCtrlIntGain" - Check the speed encoder mounting - Check the cable of the speed encoder - Exchange the speed encoder - Inform your service agency

Error number	Description
231-A920	**Error message** A920 Standstill monitoring SKERN-CC %1 **Cause of error** - SKERN-CC detected an impermissibly large axis movement in the SOS safety condition. The standstill speed (50 mm/min for feed axes or 10 rpm for spindles) was also exceeded. The maximum permissible path in the SOS condition is defined in the machine parameter "positionRangeVmin." Possible causes: - Machine parameter "positionRangeVmin" is defined too small. - The brake was deactivated before the position controller was closed. - The brake was not activated before the position controller was opened. - When an axis was switched on, some existing following error was corrected. - The brake is defective. - There was an attempt to move an axis in the SOS condition (PLC?) - The axis feed-rate enabling by the ApiToSafety datum PP_AxFeedEnable is missing. **Error correction** - Check the entry in machine parameter positionRangeVmin. - Check the sequence of deactivating the brake and closing the position controller. - Check the sequence of activating the brake and opening the position controller. - Check whether there is a following error after an axis is locked. - Check the interface signal of the SPLC PP_AxFeedEnable for the axis - Inform your service agency

Error number	Description
231-A930	**Error message** A930 Safely limited speed (SLS) exceeded %1 **Cause of error** - SKERN-CC detected an impermissibly fast axis movement above the safe reduced speed. (SLS2,SLS3,SLS4) Possible causes: - Guard door was opened (at high speed) during an axis movement. - Machine parameter for reduced speed is defined too small. **Error correction** - Check whether the guard door was opened during an axis movement. - SLS2: Check the entry in machine parameter "speedLimitSom2." - SLS3: Check the entry in machine parameter "speedLimitSom3." - SLS4: Check the entry in machine parameter "speedLimitSom4." - Inform your service agency.
231-A940	**Error message** A940 Path exceeded during restricted spindle operation %1 **Cause of error** In the limited spindle operation, the maximum permissible path of 2 revolutions was exceeded. **Error correction** - Check the path of the spindle during limited spindle operation. - Inform your service agency.

Error number	Description
231-A950	**Error message** A950 Safely limited increment (SLI) exceeded %1 **Cause of error** - The maximum permissible path was exceeded in the Safely Limited Increment (SLI) mode. - Possible causes: - The selected increment exceeds the value from the safe machine parameter distLimitJog - The drive overshoots mechanically when the limit position is reached - Drive is not optimally adjusted. **Error correction** - Enter a smaller jog increment. - Check the entry in the safe machine parameter distLimitJog. - When the jog increment end position is reached, use the control's internal oscilloscope to check the actual position value for overshooting. - Adjust the drive. - Inform your service agency.
231-AC00	**Error message** AC00 CC amplitude too high %1 CC amplitude too high %2 **Cause of error** - The amplitude of the encoder signal is too high or the signal for contamination is active. - Incorrect adjustment between head and encoder, air gap too small (exposed encoders) - Excessive supply voltage **Error correction** - Check the amplitude of the encoder signal - Inform your service agency

Error number	Description
231-AC10	**Error message** AC10 Motor encoder %1 defective C300 Zn track %1 error C310 Z1 track %1 error **Cause of error** - Motor encoder contaminated or defective - Cable defective - Encoder input defective on the control - Signal connector: Poor contact or penetration of humidity - Humidity has entered the motor - No encoder signal available - Interruption in motor encoder cable - Signal amplitude of motor encoder is missing or too small **Error correction** - Inform your service agency - Check the motor encoder connection - Check the motor encoder - Check the amplitude of the encoder signal
231-AC20	**Error message** AC20 CC frequency too high %1 CC frequency too high %2 **Cause of error** - The maximum input frequency was exceeded at an encoder input. - Noise on motor encoder signal - Vibrations on the machine **Error correction** - Inform your service agency - Check the motor encoder connection (ground connection) - Check the motor encoder - Check the encoder signal input frequency - Remove the vibrations
231-AC30	**Error message** AC30 CC ampl. too high %1 (position) **Cause of error** - The amplitude of the position encoder signal is too high - Fault in the encoder signal - Short circuit in the encoder cable - Signal amplitude of encoder is too high **Error correction** - Inform your service agency - Check the encoder connection (ground connection) - Check the encoder

Error number	Description
231-AC40	**Error message** AC40 Position encoder %1 defective **Cause of error** - Encoder contaminated - Encoder defective - Penetration of humidity - Scanning head misaligned (distance, parallelism, etc.) - Encoder cabling defective - Encoder input defective on the control **Error correction** - Inform your service agency - Check the encoder connection - Check the encoder
231-AC50	**Error message** AC50 CC freq. too high %1 (position) **Cause of error** - The maximum input frequency was exceeded at a position encoder input. - Noise on the encoder signal - Vibrations on the machine **Error correction** - Inform your service agency - Check the encoder connection (ground connection) - Check the encoder - Check the input frequency of the encoder signal. - Remove the vibrations
231-B200	**Error message** B200 CC%2 No brake test was conducted %2 **Cause of error** - MC makes no test of the motor brake(s) although it is configured by machine parameter setting. - Cannot enable the motor-brake test(s) - An axis was deselected by PLC module and the corresponding machine parameter for running a brake test for this axis is still set. **Error correction** - Check the machine parameter for conducting a brake test - Generate the service files and notify the Service Department

Error number	Description
231-B300	**Error message** B300 CC%2 No brake line test was conducted %1 **Cause of error** - MC (A channel) makes no test of the brake line, although the machine parameter setting requires it. - Timeout when calling for test of the brake control by the MC. - An axis was deselected by PLC module, although the corresponding machine parameter for running the brake line test is still set. **Error correction** - Check the machine parameter for testing the brake control or brake test - Generate the service files and notify the Service Department
231-B800	**Error message** B800 CC%2 safe inputs %1 not equal CC%2 safe inputs %1 not equal **Cause of error** - A safety-related input of the CC is longer than 400 ms unequal to MC - Different levels at the safety module input: 4 = Acknowledgement of switch-off 8 = Safe reduced velocity of axes/spindle 10 = Safe reduced velocity of auxiliary axes 11 = Operating mode 3(detachable-key switch 1, Pos3) (safe controlled stop of axes/spindle) 18 = Operating mode 2 (detachable-key switch 1,Pos2) 19 = Operating mode 4 (detachable-key switch 2) - Wiring error X65, X66, (X67) – Safety module defective **Error correction** – Check the wiring X65, X66, (X67) – Exchange the safety module - Generate the service files and notify the Service Department

Error number	Description
231-B900	**Error message** B900 CC%2 supply voltage %1 **Cause of error** - The supply voltage Vcc(x) was out of range. - +4 = undervoltage Vcc(+5 V) The load from external components (e.g. encoders) is too large. - +6 = overvoltage Vcc(+5 V) The power supply unit is defective. - +14 = undervoltage Vcc(+15 V) The power supply unit is defective. - +16 = overvoltage Vcc(+15 V) The power supply unit is defective. - -14 = undervoltage Vcc(-15 V) The power supply unit is defective. - -16 = overvoltage Vcc(-15 V) The power supply unit is defective. **Error correction** - Inform your service agency. - Measure the supply voltage Vcc(x). - Vcc(+5 V) < +4.75 V Check the encoder connections. - Vcc(+5 V) > +5.50 V Exchange the power supply unit. - Vcc(+15 V) < +14.25 V Exchange the power supply unit. - Vcc(+15 V) > +16.50 V Exchange the power supply unit. - Vcc(-15 V) < -14.25 V Exchange the power supply unit. - Vcc(-15 V) > -16.50 V Exchange the power supply unit.
231-BA00	**Error message** BA00 CC%2 operating temperature %1 **Cause of error** - Temperature inside the LE was out of permissible range. (-128... 0...+127 = measured temperature value [°C]) - Temperature sensor on board is defective. - The electrical-cabinet ventilation is insufficient (fan defective). - The ambient temperature is too high or too low. **Error correction** - Check the ventilation. - Inform your service agency.
231-BB00	**Error message** BB00 CC%2 MC command %1 to CC software is not allowed **Cause of error** - MC command not permitted for this software variant of the CC - MC command not permitted at this point in time. Decimal code = low byte of the command code (0...255) - Internal software error **Error correction** - Check the software version - Inform your service agency

Error number	Description
231-C000	**Error message** C000 No data exchange with MC **Cause of error** - Communication to the MC was interrupted. - Internal software error. **Error correction** - Inform your service agency. - Check the software version.
231-C001	**Error message** C001 Undefined error Undefined error **Cause of error** Internal software error **Error correction** - Inform your service agency - Check the software version
231-C002	**Error message** C002 MC command invalid **Cause of error** Internal software error **Error correction** - Inform your service agency - Check the software verion
231-C003	**Error message** C003 System clock MC not = CC%2 System clock MC not = CC%2 **Cause of error** - Hardware error (crystal generator) - Internal software error **Error correction** - Inform your service agency - Exchange the drive control board or processor board - Check the software version
231-C005	**Error message** C005 CC hardware is not supported **Cause of error** - The hardware version of the CC controller unit is not supported by installed the NC software - Power supply module (UV/UVR) interferes with I2C bus **Error correction** - Check the NC software version - Check/exchange the X69 ribbon cable - Exchange the power supply module (UV/UVR) - Inform your service agency

Error number	Description
231-C006	**Error message** C006 I-CTRL communication: TIME C008 I-CTRL communication: QUEUE **Cause of error** - Communication error between speed and current controller. **Error correction** - Inform your service agency. - Check the software version.
231-C007	**Error message** C007 DC-link voltage too low **Cause of error** - Line power interrupted - Inverter defective **Error correction** - Check the line power supply - Inform your service agency - Check the inverter
231-C009	**Error message** C009 Stack overflow **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version
231-C00A	**Error message** C00A PWM triangular signal error **Cause of error** - Hardware error: Triangular signal does not oscillate, or it oscillates with incorrect frequency **Error correction** - Inform your service agency - Exchange the drive control board
231-C00B	**Error message** C00B Too little main memory **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version

Error number	Description
231-C00C	**Error message** C00C LSV2, incorrect number of data **Cause of error** - The number of LSV2 data to be read is incorrect - Internal software error **Error correction** - Inform your service agency - Check the software version
231-C00D	**Error message** C00D CC%2 Checksum error in the DSP program code **Cause of error** - A checksum error was discovered in the program code of the CC controller unit - Defective CC controller unit **Error correction** - Exchange the hardware - Inform your service agency
231-C00E	**Error message** C00E Controller software timeout **Cause of error** - Internal software or hardware error **Error correction** - Inform your service agency - Check the software version - Exchange the drive control board
231-C00F	**Error message** C00F Error in software timer **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version
231-C010	**Error message** C010 Bus error in speed controller **Cause of error** - Access violation on controller periphery. **Error correction** - Inform your service agency. - Exchange the controller board.

Error number	Description
231-C011	**Error message** C011 Softw. synchronization err. **Cause of error** - Missing hardware interrupt after DSP start (>900[ms]) - Missing synchronization command of the MC before drive switch-on - Hardware is defective (MC or CC) **Error correction** - Exchange the hardware (MC or CC) - Inform your service agency.
231-C012	**Error message** C012 Pos. control cyc. time err. **Cause of error** - MC is providing erroneous cycle time for CC position controller - Hardware error **Error correction** - Inform your service agency - Check the configuration datum ipoCycle - Exchange the drive control board
231-C013	**Error message** C013 PWM frequency error **Cause of error** - The PWM frequency entered in CfgPowerStage->ampPwm-Freq is outside the permissible input range - The selected PWM frequencies must not be combined **Error correction** - Inform your service agency - Check CfgPowerStage->ampPwmFreq
231-C014	**Error message** C014 Interpolator, PWM invalid **Cause of error** - Invalid relation between interpolator clock pulse and PWM frequency. **Error correction** - Inform your service agency. - Change the ratio of interpolator clock pulse to PWM frequency. - For possible ratios see Technical Manual.

Error number	Description
231-C015	**Error message** C015 Interpolator, PWM changed **Cause of error** - Interpolator clock pulse or PWM frequency was changed. **Error correction** - Restart the control.
231-C016	**Error message** C016 Double speed not possible **Cause of error** - Control loop on X51 or X52 is defined as 'double speed', although the control loop on X53 or X54 is active - Control loop on X55 or X56 is defined as 'double speed', although the control loop on X57 or X58 is active (only CC 4xx with 8 control loops) **Error correction** - Inform your service agency - Define the control loop on X51 or X52 as 'single speed', or deactivate the PWM output X53 or X54 - Define the control loop on X55 or X56 as 'single speed', or deactivate the PWM output X57 or X58 (only CC 4xx with 8 control loops)
231-C017	**Error message** C017 PWM frequency too high **Cause of error** - For a single-speed control loop, in configuration datum ampPwmFreq the double PWM basic frequency, and in iCtrlPwmType one-half of the current controller cycle time has been set. **Error correction** - Inform your service agency - Check the configuration datum ampPwmFreq and iCtrlP-wmType - Use a double-speed control loop instead of single-speed
231-C018	**Error message** C018 Master-slave torque: Axis assignment incorrect **Cause of error** - The axis in master-slave torque control are permissible only at X15/X17 or X16/X18. **Error correction** - Inform your service agency. - Change the axis assignment.

Error number	Description
231-C020	**Error message** C020 Faulty Include file **Cause of error** - The MC and CC software were not compiled with the same Include file. **Error correction** - Check the software version and reload if necessary - Inform your service agency
231-C021	**Error message** C021 Wrong DSP version **Cause of error** - The MC and CC software were not compiled with the same Include file. **Error correction** - Inform your service agency - Check the software version and reload if necessary
231-C022	**Error message** C022 SMB or SPL configuration error **Cause of error** - Configuration error in the HSCI system - Configuration error in the safe machine operating panel MB 6xx S or a safe PL 6xxx S - A new device type has been connected that is not yet supported by the current CC software - Error in the MC's configuration data to the CC **Error correction** - Software update - Inform your service agency
231-C023	**Error message** C023 IRQ stack overflow **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version
231-C025	**Error message** C025 CC-CC communication, CRC error **Cause of error** - An HSCI station causes an offset error - Incorrect message size by MC **Error correction** - Software update - Inform your service agency

Error number	Description
231-C026	**Error message** C026 CC-CC communication, watchdog error **Cause of error** - An HSCI station causes an offset error - Incorrect message size by MC - Message transmission canceled **Error correction** - Software update - Exchange the PL 6xxx S - Inform your service agency
231-C027	**Error message** C027 HSCI message is missing **Cause of error** - HSCI message (low-prio) is missing - An HSCI station causes an error - Message list in MC is faulty **Error correction** - Check the HSCI devices - Software update - Inform your service agency
231-C028	**Error message** C028 MC acknowledgment is missing **Cause of error** - An HSCI message from the CC to the MC was not acknowledged **Error correction** - Software update - Inform your service agency
231-C02B	**Error message** C02B Watchdog error for machine operating panel **Cause of error** - Watchdog of the MB 6xx S was not retriggered - Hardware error on the MB 6xx S **Error correction** - Exchange the MB 6xx S - Inform your service agency
231-C02C	**Error message** C02C Watchdog error for PL / SPL assembly **Cause of error** - Watchdog of the PL 6xxx (S) was not retriggered - Hardware error on the PL 6xxx (S) **Error correction** - Exchange the PL 6xxx (S) - Inform your service agency

Error number	Description
231-C02D	**Error message** C02D Watchdog error, HSCI module **Cause of error** - HSCI module of the CC is defective -> HSCI messages cannot be received any longer -> The watchdog is not retriggered any longer - HSCI cable defective **Error correction** - Check the connection of the HSCI cable - Check/replace the HSCI cable - Exchange the CC - Inform your service agency
231-C02E	**Error message** C02E CC firmware update required **Cause of error** - An update of the firmware is required because of a hardware or software exchange **Error correction** - Inform your service agency
231-C02F	**Error message** C02F Error during control start up **Cause of error** - Internal software error in the MC, CC or an HSCI participant (RunUp) **Error correction** - Inform your service agency
231-C030	**Error message** C030 Alarm with supply voltages CC%2 **Cause of error** - The internal supply voltages of the CC are outside of the specified range. Please pay attention to the diagnostic message "0xC038 voltage monitoring"! **Error correction** - Check the supply voltages on the CC: - Check the wiring of X69 - Cable length at X69 within specification? - Exchange the cable on X69 - Exchange the hardware - Inform your service agency

Error number	Description
231-C031	**Error message** C031 Alarm with supply voltages Supply voltage missing at device **Cause of error** The supply voltages on a device in the HSCI line are outside of the specified range. The HSCI bus diagnosis indicates which HSCI component triggered the error. Possible devices: - MC main computer - PL inputs/outputs - MB machine operating panel - Other CCs in the HSCI chain Possible causes: - Insufficient power supply to the devices - Short circuit in the power supply - Short circuit in PL inputs and outputs **Error correction** - Check the supply voltage in the connected devices - Check the wiring for possible short circuits (e.g. PLC inputs or outputs) - Exchange the hardware - Inform your service agency
231-C032	**Error message** C032 System clock of the MC less than CC%2 **Cause of error** - MC main computer or CC controller unit is defective. **Error correction** - Inform your service agency.
231-C033	**Error message** C033 System clock of the MC greater than CC%2 **Cause of error** - MC main computer or CC controller unit is defective. **Error correction** - Inform your service agency.
231-C034	**Error message** C034 CC%2 self test in S status faulty **Cause of error** - The MC did not correctly end the the self text of the S status signals. - Internal software error **Error correction** - Check the software version - Inform your service agency

Error number	Description
231-C035	**Error message** C035 CC%2 S status test: invalid test sequence **Cause of error** - During the S status test, the MC did not follow the operational sequence of this test - Internal MC software error **Error correction** - Inform your service agency
231-C036	**Error message** C036 CC%2 S status test: invalid signal **Cause of error** - During the S status test, the MC requested an unknown or unsupported signal - Internal MC software error **Error correction** - Inform your service agency
231-C037	**Error message** C037 CC%2 S status test with drives switched on **Cause of error** - The MC requested an S status test while axes are in closed-loop control - The PLC program initiated the self-test of the control (PLC Module 9144) while axes are still in closed-loop control. - Internal MC software error **Error correction** - Check the PLC program and correct if necessary. - Inform your service agency

Error number	Description
231-C038	**Error message** C038 Voltage monitoring CC%2 voltage ID: %4 **Cause of error** - The voltage monitoring of the CC supply voltage reports an error in the CC displayed. - The cause of the error message could be the 5 V power supply of the CCs over the power bus (X69). With large line lengths on the power bus, the 5 V power supply may have to be wired additionally over X74. (Use short line lengths and large line cross section, and check voltage drops on lines between X74.) - Defective power supply unit in the supply module (5 V power pack in the UV). **Error correction** - Establish a 5 V supply between the supply module and the CCs via X74 - Check the 5 V power supply (usually on supply module X74) - Check the 5 V power supply on all CCs (X74/CC) - Check the supply bus (X69) - Check the wiring: - Wiring of the supply bus (X69) - Wiring of the 5 V supply (X74) - Check the cable length of the supply bus (X69), if necessary use double lines - Check the 5 V supply voltage at X74 of all CCs if necessary, increase the wire cross sections or reduce the cable length - Check the voltage attenuation on the cable between X74 on the supply module and X74 on the CCs - Exchange the power supply - Exchange defective hardware (CC) - Inform your service agency, stating the alarm number and Voltage ID
231-C039	**Error message** C039 Hardware error CC%2 assembly ID: %4 **Cause of error** - Hardware error found on the CC controller unit. **Error correction** - Exchange the defective hardware/CC - Read out the log - Inform your service agency

Error number	Description
231-C110	**Error message** C110 Unknown motor type %1 **Cause of error** - Error in motor table. - Internal software error. **Error correction** - Inform your service agency. - Check motor table. - Check the software version.
231-C140	**Error message** C140 Pole pair no. too large %1 **Cause of error** - Incorrect entry in motor table **Error correction** - Inform your service agency - Check the motor table
231-C150	**Error message** C150 Field current error %1 **Cause of error** - Incorrect entry in motor table **Error correction** - Inform your service agency - Check the motor table
231-C160	**Error message** C160 Grating per. motor enc. %1 **Cause of error** - Measured grating period does not agree with the entry in the motor table **Error correction** - Inform your service agency - Check the motor table (line count) - Check the motor
231-C170	**Error message** C170 Rotor time constant err. %1 **Cause of error** - The rotor time constant calculated from the rotor table is invalid **Error correction** - Inform your service agency - Check the motor table

Error number	Description
231-C180	**Error message** C180 Rated speed error %1 **Cause of error** - Incorrect entry in motor table **Error correction** - Inform your service agency - Check the motor table
231-C1D0	**Error message** C1D0 Current sensor voltage %1 **Cause of error** - Incorrect entry in power module **Error correction** - Inform your service agency - Check the power module table
231-C1E0	**Error message** C1E0 Imax of power module %1 **Cause of error** - Incorrect entry in power module table **Error correction** - Inform your service agency - Check the power module table
231-C210	**Error message** C210 Tmax of motor table %1 **Cause of error** - Incorrect temperature entry in motor table **Error correction** - Inform your service agency - Check the motor table
231-C240	**Error message** C240 Rated I of power module %1 **Cause of error** - Incorrect entry in power module table **Error correction** - Inform your service agency - Check the power module table

Error number	Description
231-C250	**Error message** C250 Rated I of motor %1 **Cause of error** - Incorrect entry in motor table **Error correction** - Inform your service agency - Check the motor table
231-C260	**Error message** C260 Imax of motor %1 error **Cause of error** - Incorrect entry in motor table **Error correction** - Inform your service agency - Check the motor table
231-C270	**Error message** C270 Nmax of motor %1 error **Cause of error** - Incorrect entry in motor table **Error correction** - Inform your service agency - Check the motor table
231-C280	**Error message** C280 Field angle %1 error **Cause of error** Incorrect entry in CfgServoMotor->MotFieldAngleAdapStartSpeed or CfgServoMotor->MotFieldAngleAdapMaxAngle **Error correction** - Inform your service agency. Check CfgServoMotor->MotFieldAngleAdapStartSpeed or CfgServoMotor->MotFieldAngleAdapMaxAngle.
231-C290	**Error message** C290 Uz %1 error **Cause of error** - Incorrect entry in CfgPowerStage->ampBusVoltage (dc-link voltage Uz) **Error correction** - Inform your service agency. - Check the entry CfgPowerStage->ampBusVoltage.

Error number	Description
231-C2A0	**Error message** C2A0 Encoder input %1 **Cause of error** Incorrect entry in CfgAxisHardware->selectEncoderIn (speed encoder). Internal software error. **Error correction** Inform your service agency. Check the entry in CfgAxisHardware->selectEncoderIn. Check the software version.
231-C2B0	**Error message** C2B0 PWM output %1 **Cause of error** - Incorrect entry in parameter "CfgAxisHardware->analogOutput" (nominal speed value output). - Internal software error. **Error correction** - Internal software error. - Check parameter "CfgAxisHardware->analogOutput". - Check the software version.
231-C2C0	**Error message** C2C0 Band-pass parameter %1 **Cause of error** - Incorrect entry in CfgSpeedControl->vCtrlFiltDamping1. - Internal software error. **Error correction** - Inform your service agency. - Check the entry in CfgSpeedControl->vCtrlFiltDamping1. - Check the software version.
231-C2D0	**Error message** C2D0 Encoder line count %1 **Cause of error** - Encoder line count was changed **Error correction** - Restart the control
231-C2E0	**Error message** C2E0 Motor pole pair number %1 **Cause of error** - Motor pole pair number was changed **Error correction** - Restart the control

Error number	Description
231-C2F0	**Error message** C2F0 DIR in motor table %1 **Cause of error** - DIR in the motor table was changed **Error correction** - Restart the control
231-C330	**Error message** C330 Motor temp. too high %1 **Cause of error** - Motor encoder cable defective - Temperature sensor defective - Signal connector: Poor contact or penetration of humidity - Humidity has entered the motor **Error correction** - Let the motor cool down - Inform your service agency - Check the motor encoder cable - Measure the temperature sensor
231-C340	**Error message** C340 Unknown counter component %1 **Cause of error** - Parameter error at the active axis in machine parameter speedEncoderInput - Hardware defective (CC) - Motor encoder defective - Incorrect software version **Error correction** - Check the machine parameter (speedEncoderInput) - Check the software version - Operate the motor at another encoder input - Exchange the drive control board - Inform your service agency

Error number	Description
231-C350	**Error message** C350 Axis module %1 not ready **Cause of error** - Safety relay not on (e.g. connectors X71 and X72 of the UV, X73 of the HEIDENHAIN expansion board for Simodrive) - PWM bus cable interrupted - Interruption in the electrical cabinet - Defective axis module - PWM interface on the control defective - No pulse release for the axis module - Uz too high - 5V power supply too weak - Inverter not ready - Motor control board defective - PWM cable defective - Noise signals **Error correction** - Inform your service agency - Check the control and cabling of the pulse release - Check Uz - If the power supply is not regenerative, is the braking resistor connected? - If the power supply is regenerative, is the energy recovery activated? - Check the cable ground and shield - Exchange the power module - For P controls, exchange the interface card - Exchange the motor control board
231-C370	**Error message** C370 Angle error motor encdr. %1 **Cause of error** - Motor encoder defective - Motor encoder exchanged - Motor encoder cable defective - Signal connector: Poor contact or penetration of humidity - Humidity has entered the motor - Drive control board defective **Error correction** - Run a new field orientation after encoder exchange - Inform your service agency - Check motor encoder and leads - Exchange the drive control board

Error number	Description
231-C380	**Error message** C380 Motor %1 not controllable **Cause of error** - Motor encoder cable defective - Motor defective - I2t monitoring has responded - Signal connector: Poor contact or penetration of humidity - Humidity has entered the motor - Motor brake is on **Error correction** - Check the motor cabling - Inform your service agency - Check the motor and the motor encoder cabling - Check the motor table entry - Check I2t monitoring
231-C390	**Error message** C390 Error in 3-D touch probe %1 **Cause of error** - Internal software error - Hardware error on drive control board **Error correction** - Check the software version - Exchange the drive control board - Inform your service agency
231-C3A0	**Error message** C3A0 Incorrect ref. position %1 **Cause of error** - Incorrect motor selected - Grounding error on motor encoder cable (disturbance on reference signal line) - Motor encoder defective **Error correction** - Inform your service agency - Check the motor selection - Check the cabling of the motor encoder (grounding) - Exchange the motor

Error number	Description
231-C3B0	**Error message** C3B0 Motor %1 does not run under max. current **Cause of error** - Motor locked or blocked - Inverter defective - Motor defective - Wrong motor addressed - Assignment of PWM outputs entered incorrectly - Motor power cables mismatched - Motor encoder cable mismatched - Incorrect motor connection - The motor is loaded with the maximum torque **Error correction** - Check the inverter and exchange it if necessary - Inspect the motor and cabling - Check the motor load - Check whether the shaft can run freely - Check the machine parameters - Inform your service agency
231-C3C0	**Error message** C3C0 Motor current %1 too high **Cause of error** - Incorrect current controller parameters - Incorrect parameters in the motor table - Power module defective - Motor cable defective - Motor defective - Humidity has entered the motor - Motor control board defective **Error correction** - Inform your service agency - Are the correct motor and power module selected? - Check the current control adjustment - Check the motor and motor cable for a short circuit - Exchange power module or drive control board
231-C3D0	**Error message** C3D0 PWM component defective %1 **Cause of error** - Internal hardware error **Error correction** - Inform your service agency - Exchange the drive control board

Error number	Description
231-C3E0	**Error message** C3E0 Err. in rated U of motor %1 **Cause of error** - Motor rated voltage outside of permitted input range **Error correction** - Inform your service agency - Check the entry in the motor table
231-C3F0	**Error message** C3F0 EnDat not found %1 **Cause of error** - EnDat communication error - Motor encoder contaminated or defective - Signal cable defective - Encoder input defective on the control - Signal connector: Poor contact or penetration of humidity - Humidity has entered the motor **Error correction** - Inform your service agency - Check the motor table (SYS column) - Exchange the motor control board (or better, the control) - Check the speed encoder cable (defective or too long) - Check the speed encoder - Check the cable ground and shield
231-C400	**Error message** C400 Line count incorrect %1 **Cause of error** - Line count from the motor table does not match the downloaded values **Error correction** - Inform your service agency - Check configuration data for linear distance of one motor revolution and distance for the number of signal periods. - Check the motor table (columns TYPE and STR). - Check the speed encoder

Error number	Description
231-C410	**Error message** C410 Rotor position %1 undefined **Cause of error** - Signal cable defective - Motor encoder contaminated or defective - Encoder input defective on the control - Signal connector: Poor contact or penetration of humidity-Humidity has entered the motor - Motor control board defective **Error correction** - Inform your service agency - Exchange the motor - Check the speed encoder cable - Exchange the motor control board (or better, the control)
231-C420	**Error message** C420 Uncontrollability %1 caused by incorrect parameters **Cause of error** - Feedforward-control parameters are set incorrectly (acceleration, friction) - Excessive acceleration - Controller parameters are set incorrectly (Ki, Kp, Kd) - Filters set incorrectly (band rejection, low pass) - Inverter is defective (IGBT) - Incorrect motor selected in motor table **Error correction** - Inform your service agency - Check the adjustment of the axes - Check the inverter
231-C430	**Error message** C430 Error of position input %1 **Cause of error** - Position encoder input does not exist. - Position encoder input is not correctly connected. - Position encoder input is defective. **Error correction** - Inform your service agency. - Install the position encoder input. - Check the connection of the position encoder input. - Exchange the position encoder input.

Error number	Description
231-C440	**Error message** C440 PWM frequency %1 incorrect **Cause of error** - PWM frequency within a control group is incorrect **Error correction** - Inform your service agency - Check configuration data (PWM frequency) - PWM frequency > 5000 Hz only with suitable hardware and only with PWM outputs X51, X52, X57 and X58. - PWM frequency <= 5000Hz must be identical within the control groups. - PWM frequency > 3200 Hz
231-C450	**Error message** C450 Wrong encoder %1 **Cause of error** - Incorrect entry in motor table column SYS. - Speed encoder cable defective. - Speed encoder defective. - Motor control board defective. **Error correction** - Inform your service agency. - Check the motor table (column SYS). - Check the motor encoder cable. - Exchange the motor. - Exchange the motor control board.
231-C460	**Error message** C460 Motor speed too high %1 **Cause of error** - Motor cannot be servocontrolled. **Error correction** - Inform your service agency. - Check the software version.
231-C470	**Error message** C470 No nominal speed values %1 **Cause of error** - Internal software error - Position controller cycle time too short **Error correction** - Inform your service agency - Check the software version - Check the configuration datum CfgCycleTimes->ipoCycle

Error number	Description
231-C480	**Error message** C480 V/Hz control mode with encoder %1 **Cause of error** - In the motor table, volts-per-hertz control mode is set (STR == 0), but a motor encoder is indicated (SYS <> 0) **Error correction** - Correct the encoder entry in the motor table - Inform your service agency
231-C4A0	**Error message** C4A0 Inverter %1 is not active **Cause of error** - Charging contactor and main contactor on the supply unit is not on (e.g. connector X70 on UV) - Safety relay not on (e.g. connectors X71 and X72 of the UV, X73 of the HEIDENHAIN expansion board for Simodrive) - PWM bus cable interrupted - Interruption in the electrical cabinet (unit bus, PWM ribbon cable) - Defective inverter, (supply unit and/or power modules, compact inverter) - Inverter switched off (PLC, SH1) - Inverter defective - Motor defective - Humidity has entered the motor - Incorrect motor selected in motor table - Motor power cables mismatched - Motor connected incorrectly **Error correction** - Inform your service agency - Check the inverter and wiring - Check the motor and wiring
231-C4C0	**Error message** C4C0 No motor current %1 **Cause of error** - Motor connected incorrectly or not at all (contactor) - Inverter defective - Motor defective - Incorrect motor selected in motor table - Motor power cables mismatched - DC-link voltage missing **Error correction** - Check the connection of the DC link - Check the motor and wiring - Check the inverter - Inform your service agency

Error number	Description
231-C4D0	**Error message** C4D0 Error in torque constant %1 **Cause of error** - If the value for the torque constant is 0 or >9999, it can have the following causes: 1) Motor.mot: No-load voltage and/or rated rpm have an invalid value (or 0) 2) Motor.sn: The entry for the torque constant is 0 or >9999 **Error correction** - Check the motor table
231-C4E0	**Error message** C4E0 Field angle determination %1 is not allowed in this mode **Cause of error** - The selected process for determining the field angle is invalid or impossible with this encoder. **Error correction** - Inform your service agency. - Check the entry in the SYS column of the motor table (see Technical Manual).
231-C4F0	**Error message** C4F0 Command not allowed **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version
231-C500	**Error message** C500 CfgSpeedControl->vCtrlDiffGain %1 too large **Cause of error** - Differential factor is too large (max. value 0.5 [As^2/rev]) **Error correction** - Inform your service agency
231-C510	**Error message** C510 Drive release %1 not allowed **Cause of error** - During readout of the electronic ID label the power module must not be in the "ready" status (-SH1 is inactive). **Error correction** - Inform your service agency - Check the software version

Error number	Description
231-C520	**Error message** C520 Timeout in position controller %1 C660 Timeout in position controller %1 **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version
231-C530	**Error message** C530 Timeout in speed controller %1 **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version
231-C540	**Error message** C540 Timeout in current controller %1 **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version
231-C550	**Error message** C550 Error in calculation of current %1 **Cause of error** - This is an internal software error - Incorrect parameters of the filter in the controller (e.g. bandwidth of the band-rejection filter is very large or the bandwidth = 0 at high center frequency) **Error correction** - Check the machine parameters of the filter for control - Set the bandwidth of band-rejection filter (machine parameter vCtrlFiltBandWidth) to a value unequal to 0 - Inform your service agency.

Error number	Description
231-C560	**Error message** C560 Param. filter 1 invalid %1 **Cause of error** - Incorrect entry in CfgSpeedControl->vCtrlFiltDamping1, CfgSpeedControl->vCtrlFiltFreq1 oder CfgSpeedControl->vCtrlFiltType1 - This is an internal software error **Error correction** - Inform your service agency - Check entries in CfgSpeedControl->vCtrlFiltDamping1, CfgSpeedControl->vCtrlFiltFreq1 or CfgSpeedControl->vCtrlFiltType1 - Check the software version
231-C570	**Error message** C570 Param. filter 2 invalid %1 **Cause of error** - Incorrect entry in CfgSpeedControl->vCtrlFiltDamping2, CfgSpeedControl->vCtrlFiltFreq2 or CfgSpeedControl->vCtrlFiltType2 - This is an internal software error **Error correction** - Inform your service agency - Check entries in CfgSpeedControl->vCtrlFiltDamping2, CfgSpeedControl->vCtrlFiltFreq2 or CfgSpeedControl->vCtrlFiltType2 - Check the software version
231-C580	**Error message** C580 Param. filter 3 invalid %1 **Cause of error** - Incorrect entry in CfgSpeedControl->vCtrlFiltDamping3, CfgSpeedControl->vCtrlFiltFreq3 or CfgSpeedControl->vCtrlFiltType3 - This is an internal software error **Error correction** - Inform your service agency - Check entries in CfgSpeedControl->vCtrlFiltDamping3, CfgSpeedControl->vCtrlFiltFreq3 or CfgSpeedControl->vCtrlFiltType3 - Check the software version

Error number	Description
231-C590	**Error message** C590 Param. filter 4 invalid %1 **Cause of error** - Incorrect entry in CfgSpeedControl->vCtrlFiltDamping4, CfgSpeedControl->vCtrlFiltFreq4 or CfgSpeedControl->vCtrl-FiltType4 - This is an internal software error **Error correction** - Inform your service agency - Check entries in CfgSpeedControl->vCtrlFiltDamping4, CfgSpeedControl->vCtrlFiltFreq4 or CfgSpeedControl->vCtrl-FiltType4 - Check the software version
231-C5A0	**Error message** C5A0 Param. filter 5 invalid %1 **Cause of error** - Incorrect entry in CfgSpeedControl->vCtrlFiltDamping5, CfgSpeedControl->vCtrlFiltFreq5 or CfgSpeedControl->vCtrl-FiltType5 - This is an internal software error **Error correction** - Inform your service agency - Check entries in CfgSpeedControl->vCtrlFiltDamping5, CfgSpeedControl->vCtrlFiltFreq5 or CfgSpeedControl->vCtrl-FiltType5 - Check the software version
231-C5B0	**Error message** C5B0 Illegal reference run of motor encoder %1 **Cause of error** - A touch-probe cycle is active while a reference value is requested by the motor encoder. **Error correction** - Inform your service agency
231-C5C0	**Error message** C5C0 Illegal reference run of position encoder %1 **Cause of error** - A touch-probe cycle is active while a reference value is requested by the position encoder. **Error correction** - Inform your service agency

Error number	Description
231-C5E0	**Error message** C5E0 Machine parameter compIpcJerkFact is faulty **Cause of error** - On the CC 424, the input range for "compIpcJerkFact" (following error in the jerk phase) is 0.0 to 0.5. **Error correction** - Inform your service agency - Check the input value in the parameter "compIpcJerkFact"
231-C5F0	**Error message** C5F0 Wrong position-encod. input **Cause of error** - An incorrect input was selected for the position encoder ("posEncoderInput" parameter) - Possible configurations CC424: 6 control loops: X201 to X206 8 control loops: X201 to X208 10 control loops: PWM outputs X51 to X56: X201 to X206 PWM outputs X57 to X60: X207 to X210 12 control loops: PWM outputs X51 to X56: X201 to X206 PWM outputs X59 to X64: X209 to X214 14 control loops: PWM outputs X51 to X58: X201 to X208 PWM outputs X59 to X64: X209 to X214 16 control loops: PWM outputs X51 to X58: X201 to X208 PWM outputs X59 to X66: X209 to X216 - Possible configuration CC61xx: PWM output <-> position encoder X51 <-> X201 X52 <-> X202 X53 <-> X203 X54 <-> X204 X55 <-> X205 X56 <-> X206 **Error correction** - Inform your service agency - Check the configuration datum posEncoderInput
231-C600	**Error message** C600 Current offset %1 is too large **Cause of error** - The current offset of the power stage is too large **Error correction** - Inform your service agency - Check/replace the PWM cable - Replace the power module - PWM output to CC is defective

Error number	Description
231-C610	**Error message** C610 TRC: PWM freq. too high %1 **Cause of error** - The axis for which the torque ripple compensation was activated through MP2260.x is being driven with a PWM frequency of more than 5000 Hz. **Error correction** - Inform your service agency - Adjust the PWM frequency for the axis to a value less than or equal to 5000 Hz - To deactivate the compensation, delete the entry in MP2260.x
231-C620	**Error message** C620 TRC: Invalid parameter %1 **Cause of error** - Invalid parameters in the compensation file of the axis **Error correction** - Inform your service agency - Find again the compensation parameters with TNCopt under Optimization/Torque Ripple Compensation - To deactivate the compensation, delete the entry in MP2260.x
231-C640	**Error message** C640 PIC switching not possible in %1 **Cause of error** - The nominal value(S in inverter.inv) changed - after DSP start, or - after the current controller adjustment is started, or - after the readiness of the power module is switched. **Error correction** - Check the LT entry (configuration datum ampName) in the machine configuration - Check the nominal value of the PICS (the S column in in inverter.inv) - If necessary, change the PWM frequency (to >= 5 kHz) - If necessary, replace the power module
231-C650	**Error message** C650 No ENDAT interpolation factor %1 **Cause of error** - No ENDAT interpolation factor received by MC - This is an internal software error **Error correction** - Check the software version - Inform your service agency

Error number	Description
231-C670	**Error message** C670 Motor encoder: EnDat 2.2 not possible %1 **Cause of error** - EnDat communication is defective - An encoder with EnDat 2.2 interface is selected in the motor table, although no EnDat 2.2 encoder is connected - The EnDat 2.2 protocol cannot be read - The EnDat measuring system has too low an interpolation rate in EnDat2.2 mode (less than 1024, e.g. EQN 1325). Operation possible only in EnDat2.1 mode **Error correction** - Check whether the encoder supports EnDat 2.2 - Check the motor table (SYS column) - Check the configuration datum motEncType - Check the cable ground and shield - Check the cable (compare the cable ID number with the documentation) - Check the speed encoder cable (defective or too long) - Check the speed encoder - Change EnDat mode (motEncType) - Exchange the motor control board - Inform your service agency
231-C680	**Error message** C680 Position encoder: EnDat 2.2 not possible %1 **Cause of error** - EnDat communication is defective - An encoder with EnDat 2.2 interface is selected in posEncoderType, although no EnDat 2.2 encoder is connected - The EnDat 2.2 protocol cannot be read **Error correction** - Check whether the position encoder supports EnDat 2.2 - Check the configuration datum posEncoderType - Check the cable ground and shield - Check the cable (compare the cable ID number with the documentation) - Check the position encoder cable (defective or too long) - Check the position encoder - Change EnDat mode (posEncoderType) - Exchange the motor control board - Inform your service agency
231-C690	**Error message** C690 DQ communication error %1 **Cause of error** - DRIVE-CLiQ communication is disturbed- DRIVE-CLiQ communication has been interrupted **Error correction** - Inspect the DRIVE-CLiQ cabling - Inform your service agency

Error number	Description
231-C6A0	**Error message** C6A0 Controller software timeout %1 IRQ-ID=%4 BOARD-ID= %2 18092 Controller software timeout BOARD-ID %2 18093 Controller software timeout BOARD-ID %2 **Cause of error** - The time monitor of the controller software reports an expiration. - Internal software error **Error correction** - Inform your service agency
231-C6B0	**Error message** C6B0 DQ init error %1 State=%4 ID=%5 Port=%6 **Cause of error** - DRIVE-CLiQ initialization error **Error correction** - Evaluation of additional information - Inform your service agency
231-C6C0	**Error message** C6C0 DQ PWM frequency was changed %1 **Cause of error** - With DRIVE-CLiQ devices, switching the PWM frequency with AmpPwmFreq does not become effective until after a reboot. **Error correction** - Acknowledge the error and restart the control. - Inform your service agency
231-C6D0	**Error message** C6D0 Motor %1 does not react. Power stage not ready **Cause of error** - Motor does not react. Power module is not ready - The power stage release is faulty - Inverter is defective - PWM cable is defective - Wrong motor selected - Assignment of PWM outputs entered incorrectly **Error correction** - Check the inverter and exchange it if necessary - Inspect the motor and the cabling - Check the motor load - Check whether the motor can run freely - Check the machine parameters - Inform your service agency

Error number	Description
231-C6E0	**Error message** C6E0 Amplitude test in axis %1: speed encoder is incorrect, test %4 **Cause of error** The internal test of encoder-amplitude monitoring at the speed inputs revealed an error: Test 1: The test for "Amplitude too low" failed. Test 2: The test for "Amplitude too high" failed. - The entry in pwmSignalOutput/analogOffset does not match the wiring of the speed encoders. - Incorrect entry in machine parameter motEncType (e.g. Z1-track operation selected for EnDat2.2 encoder) - Speed encoder cable is interrupted, or encoder cable is defective - Speed encoder is defective - CC controller unit is defective **Error correction** - Compare/Check the entry in MP pwmSignalOutput/analogOffset to the speed encoder cabling - Check the entry in machine parameter motEncType - Check the speed encoder cable / Exchange the cable - Exchange the speed encoder - Exchange the CC controller unit - Inform your service agency
231-C6F0	**Error message** C6F0 Amplitude test in axis %1: pos. encoder is incorrect, test %4 **Cause of error** The internal test of encoder-amplitude monitoring at the position inputs revealed an error: Test 1: The test for "Amplitude too low" failed. Test 2: The test for "Amplitude too high" failed. - The entry in posEncoderInput/analogOffset does not match the wiring of the position encoder. - Incorrect entry in machine parameter posEncodeType (e.g. analog encoder signal selected for digital encoder) - Position encoder cable is interrupted, or encoder cable is defective - Position encoder is defective - CC controller unit is defective **Error correction** - Compare/check the entry in MP posEncoderInput/analogOffset to the position encoder cabling - Check the entry in machine parameter posEncodeType - Check the position encoder cable / Exchange the cable - Exchange the position encoder - Exchange the CC controller unit - Inform your service agency

Error number	Description
231-C700	**Error message** C700 DQ-ALM: Line power failure %1 **Cause of error** - A power phase failure was detected at the DRIVE-CLiQ regenerative module ALM. - Power voltage supply is disturbed. **Error correction** - Check the protective devices of the line power voltage supply. - Check the wiring of the line power voltage supply. - Inform your service agency
231-C710	**Error message** C710 Error in drive switch-off %1 **Cause of error** - The drive could not be decelerated to 0 rpm with the time configured in machine parameter vCtrlTimeSwitchOff. - Possible causes: - An IGBT of the power module switched off. - The machine parameter vCtrlTimeSwitchOff is set incorrectly. - The permissible load was exceeded. **Error correction** - Check the machine parameter vCtrlTimeSwitchOff - Check the load - Exchange the CC controller unit - Inform your service agency
231-C720	**Error message** C720 The software is not suitable for the PLASTIC_INJECTION **Cause of error** PLASTIC_INJECTION commands are not permitted **Error correction** Inform your service agency
231-C730	**Error message** C730 CC%2 CPU0 impermissible data processing 13038 UM: CC%2 %1 CPU0 impermissible data processing **Cause of error** Internal software error: There was an attempt to access an impermissible memory area **Error correction** - Please generate a service file soon, reboot the control, and then generate another service file - Transmit both service files to the Service department for further inspections by HEIDENHAIN

Error number	Description
231-C740	**Error message** C740 CC%2 CPU1 impermissible data processing 13039 UM: CC%2 %1 CPU1 impermissible data processing **Cause of error** Internal software error: There was an attempt to access an impermissible memory area **Error correction** - Please generate a service file soon, reboot the control, and then generate another service file - Transmit both service files to the Service department for further inspections by HEIDENHAIN
231-C750	**Error message** C750 CC%2 CPU0 impermissible instruction processing 1303A UM: CC%2 %1 CPU0 impermissible instruction processing **Cause of error** Internal software error: There was an attempt to access an impermissible memory area **Error correction** - Please generate a service file soon, reboot the control, and then generate another service file - Transmit both service files to the Service department for further inspections by HEIDENHAIN
231-C760	**Error message** C760 CC%2 CPU1 impermissible instruction processing 1303B UM: CC%2 %1 CPU1 impermissible instruction processing **Cause of error** Internal software error: There was an attempt to access an impermissible memory area **Error correction** - Please generate a service file soon, reboot the control, and then generate another service file - Transmit both service files to the Service department for further inspections by HEIDENHAIN
231-C770	**Error message** C770 System errors when activating a drive% 1 **Cause of error** A drive was switched on that either - is not active, meaning it is set to inactive in the kinematics configuration, or - its parameterization has not been completed **Error correction** Inform your service agency

Error number	Description
231-C780	**Error message** C780 Error while braking a drive %1 **Cause of error** It was not possible to brake the drive when switching off. An unexpected acceleration of the drive was detected during the braking procedure. Possible causes: - Switch-off time is parameterized to be too brief - Load is too high - Too much noise in the signals of the speed encoder **Error correction** Remedy: - Check the machine parameter timeLimitStop1 (system with integrated functional safety (FS)) or delayTimeSTOatSS1 (system with external safety) - Check the load - Check the speed encoder - Inform your service agency
231-CFF0	**Error message** CFF0 Alarm axis %1 module=%4 line=%5 10022 CC %2 axis %1: Measurement was not finished correctly. 180A5 Axis %1: error occurred during measurement **Cause of error** - This is an internal software error **Error correction** - Inform your service agency
231-D000	**Error message** D000 CC%2 DP RAM area %1 **Cause of error** - Internal software error 1...255 = area number **Error correction** - Inform your service agency. - Check the software version.
231-D100	**Error message** D100 CC%2 software error %1 **Cause of error** - Internal software error 0...255 = code for error in software module/routine **Error correction** - Inform your service agency. - Check the software version.

Error number	Description
231-D300	**Error message** D300 Data transm. SPL to CC, CRC error at HSCI addr.: %1 **Cause of error** - HSCI connecting cable is defective or not connected - Data transmission error in the HSCI system - A safe PL 6xxx FS is transmitting erroneous data - Incorrect HSCI message size (MC software) **Error correction** - Check the HSCI connecting cable - Exchange the safe PL 6xxx FS - Software update - Inform your service agency
231-D400	**Error message** D400 Data transm. SPL to CC, watchdog error at HSCI addr.: %1 **Cause of error** - Data transmission error in the HSCI system - A safe PL 6xxx FS is transmitting erroneous data - The HSCI transmission is disturbed **Error correction** - Software update - Exchange the PL 6xxx FS - Inform your service agency
231-E000	**Error message** E000 Start test of the cut-out channels not possible **Cause of error** - Start of the "tests of cutout channels" by PLC not possible because the control is not in the "AUTO" condition. - The motors of at least one axis group are not yet switched off. **Error correction** - Check the PLC program and correct if necessary. - Inform your service agency.
231-E001	**Error message** E001 Status NR1/NR2 not equal **Cause of error** - NR2 input incorrectly connected - Internal software error **Error correction** - Inform your service agency - Check the wiring - Check the software version

Error number	Description
231-E002	**Error message** E002 Status em.stop input signal not equal to test output: T2 **Cause of error** - The signal level of the test output 'T2' differs from that of the 'Emergency Stop' input of the CC. CC424: '-NE2' signal CC61xx: '-ES.B' signal **Error correction** - Check the wiring - Inform your service agency
231-E003	**Error message** E003 PLC module 9169 illegal **Cause of error** - PLC module 9169 in safety-oriented software (illegal) - Software error **Error correction** - Inform your service agency - Check the PLC program - Check the software version
231-E004	**Error message** E004 SH1 status test on active **Cause of error** - The measured status of the '-SH1' signal is 'high' level - The '-SH1' signal does not change to the 'active' ('low' level) status, although the MC is no longer triggering the corresponding watchdog. **Error correction** - Internal software error - Hardware defective - Inform your service agency
231-E005	**Error message** E005 SH1 status test on inactive **Cause of error** - The measured status of the '-SH1' signal is 'low' level - The '-SH1' signal does not change to the 'inactive' ('high' level) status, although the MC is triggering the corresponding watchdog. **Error correction** - Internal software error - Hardware defective - Inform your service agency

Error number	Description
231-E006	**Error message** E006 CC%2 input (NE2) not equal to 0 **Cause of error** Error in dynamic test of the 2nd emergency-stop loop. In the dynamic test, O V is expected at input NE2 after no later than 1.5 min. If a voltage of 24 V is present here, this error message appears. **Error correction** - Inform your service agency - Check the wiring - Check the emergency-stop button - Exchange the hardware
231-E007	**Error message** E007 CC%2 S input not equal 0 **Cause of error** Error in dynamic test of the 2nd emergency-stop loop. In the dynamic test, O V is expected at all door contacts and key-operated-switch inputs after no later than 1.5 min. If a voltage of 24 V is present here, this error message appears. **Error correction** - Inform your service agency - Check the wiring - Check the door contacts, key-operated switches - Exchange the hardware
231-E008	**Error message** E008 SRG speed too high **Cause of error** - Safe reduced rotational velocity (SRG) was exceeded - No standstill in safe controlled stop (SBH) operating mode **Error correction** Inform your service agency
231-E009	**Error message** E009 Incorrect gear range **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version

Error number	Description
231-E00A	**Error message** E00A Safe machine param. error **Cause of error** - Incorrect CRC checksum found and transferred over SG parameter memory - Communication error MC <-> CC - Wrong CC software - Hardware defect (memory fault) - Software error **Error correction** - Inform your service agency
231-E00B	**Error message** E00B Cutout channels test error **Cause of error** - Illegal code received for conducting the test. **Error correction** - Inform your service agency - Internal software error
231-E00C	**Error message** E00C Error in parameter transfer **Cause of error** - Incorrect parameter for analog spindle. - Software version MC. **Error correction** - Inform your service agency. - Check the parameter. - Check the software version.
231-E00D	**Error message** E00D Error in parameter transfer **Cause of error** - Incorrect parameter for analog spindle. - Software version MC. **Error correction** - Inform your service agency. - Check the parameter. - Check the software version.
231-E00E	**Error message** E00E Illegal pulse deletion test **Cause of error** - Command for pulse deletion test was received, although the previous test is not yet finished. **Error correction** - Inform your service agency - Internal software error

Error number	Description
231-E00F	**Error message** E00F Brake test not performed **Cause of error** - MC runs no test of the motor brake(s) although it is necessary according to parameter settings. - MC does not run the test of the motor brake(s) within 2 seconds. **Error correction** - Inform your service agency
231-E010	**Error message** E010 SH2 status test on active **Cause of error** - The measured status of the '-SH2' signal is 'high' level - The '-SH2' signal does not change to the 'active' ('low' level) status, although the CC is no longer triggering the corresponding watchdog. **Error correction** - Internal software error - Hardware defective - Inform your service agency
231-E011	**Error message** E011 SH2 status test on inactive **Cause of error** - The measured status of the '-SH2' signal is 'low' level - The '-SH2' signal does not change to the 'inactive' ('high' level) status, although the CC is triggering the corresponding watchdog. **Error correction** - Internal software error - Hardware defective - Inform your service agency
231-E012	**Error message** E012 N0 status test on active **Cause of error** - The measured status of the '-N0' signal is 'high' level - The '-N0' signal does not change to the 'active' ('low' level) status **Error correction** - Software error - Hardware defective - Inform your service agency

Error number	Description
231-E013	**Error message** E013 N0 status test on inactive **Cause of error** - CCU shut-off signal: -N0 does not switch to high level - The measured status of the '-N0' signal is 'low' level **Error correction** - Software error - Hardware defective - Inform your service agency
231-E014	**Error message** E014 Err. during -N0 signal test **Cause of error** - Change in the level of the CC1 switch-off signal: -N0 is not recognized by the CC0. - Internal software error - Hardware error **Error correction** - Check the software version - Inform your service agency
231-E015	**Error message** E015 CC protective door open in brake test **Cause of error** - The protective doors of the work envelope were opened during the brake test. **Error correction** - Close the protective doors. - Switch off and then restart the control. - The switch-off test and brake test will be started automatically.

Error number	Description
231-E018	**Error message** E018 CC%2 SPLC alarm ERR-ID=%4 SST0=%5 SST1=%6 OUT=%7 **Cause of error** - Error in the SPLC run-time program Meaning of the alarm ID 100, 101, 102, 104 and 200 to 206: Internal software error Meaning of alarm ID 103: The value of the read-back output with the number "OUT" is "1", although "0" was commanded by the SPLC for this output. **Error correction** - Measure for alarm ID 100, 101, 102, 104 and 200 to 206: Please inform your service agency - Measure for alarm ID 103: Check whether the output with the number "OUT" has a short circuit with +24 V. If this is not the case, please inform your service agency. - Inform your service agency.

Error number	Description
231-E019	**Error message** E019 CC%2 SPLC configuration error ERR-ID=%4 **Cause of error** Alarm ID 1: - Internal software error Alarm ID 2: - Invalid value of machine parameter CfgSafety.inverseInputNoA, CfgSafety.inverseInputNoB, or CfgSafety.testInputNo Alarm ID 3: - The version of the NC software was changed after the safety inspection and acceptance of the machine. - The version of the NC software and the version of the SplcApiMarker.def file being used that is stored in the safety-related machine parameter CfgSafety.s- plcApiVersion do not match. Alarm ID 4: - Permitted number of SPLC markers (1000/2000) was exceeded. Alarm ID 5: - Permitted number of SPLC DWORDs (1000/3000) was exceeded. Alarm ID 6: - Permitted number of PlcToSPLC transfer markers (64) was exceeded. Alarm ID 7: - Permitted number of PlcToSPLC transfer DWORDs (32) was exceeded. Alarm ID 8: - Marker transfer area PlcToSPLC is faulty. Alarm ID 9: - DWORDs transfer area PlcToSPLC is faulty. **Error correction** Alarm ID 1: - Inform your service agency Alarm ID 2: - Check the values of the machine parameters CfgSafety.inverseInputNoA, CfgSafety.inverseInputNoB, and CfgSafety.testInputNo. Alarm ID 3: - Transfer the 'SplcApiMarker.def' file appropriate to the installed NC software version into the SPLC project. - Enter the value of the constant SPLC_API_VERSION from this file into the safety-related machine parameter CfgSafety.splcApiVersion. - Repeat the safety inspection and acceptance of the machine with the appropriate comprehensiveness. Alarm ID 4: - Check the SPLC program and CfgPlcSafety.splcMarkers Alarm ID 5: - Check the SPLC program and CfgPlcSafety.splcDWords Alarm ID 6 and 8:

Error number	Description
	- Check the SPLC program and CfgPlcSafety.splcMarkers-FromPlc[] Alarm ID 7 and 9: - Check the SPLC program and CfgPlcSafety.splcDWords-FromPlc[]

Error number	Description
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 200 - Machine parameter axisGroup: Assigned axis group out of range 201 - Machine parameter axisGroup: Entered axis group not in use. 202 - Drive type and axis group type do not match. 210 - Permissible number of spindles was exceeded 300 - Machine parameter pwmSignalOutput. Input range exceeded 301 - Machine parameter pwmSignalOutput. The same value was used in two separate entries 400 - Machine parameter timeLimitStop1: Input range exceeded 401 - Machine parameter timeLimitStop2: Input range exceeded 600 - Machine parameter distPerMotorTurn: Input range exceeded 700 - Machine parameter speedLimitSom2, speedLimitSom3, speedLimitSom4, rpmLimitSom2, rpmLimitSom3, rpmLimitSom4: Permissible velocity limit exceeded 800 - An unknown additional function is activated in the config object CfgSafety 810 - Machine parameter timeToEmStopTest: Input range exceeded 820 - Machine parameter watchdogTime: Input range exceeded 900 - Machine parameter brakeAfter: Entered connective operation is not allowed 901 - Machine parameter brakeAfter: A connective operation to yourself is not allowed 1000 - Machine parameters plcCount: PLC / SPLC-MC input range of configured cycle time has been exceeded 1100 - Machine parameters idleState: Configuration of subsequent condition after a stop reaction SS2 is out of the permissible range 1200 - The drive assignment is missing for a safe axis - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 200 - Check the entered value of the axis group 201 - Enter only axis groups that are actually being used 202 - Check the axis group assignment 210 - Check the number of spindles in your system 300 - Check the entity pwmSignalOutput. Input value too large. 301 - Check entity pwmSignalOutput. The same value is in

Error number	Description
	two separate entries. 400 - Check the entered time Parameter value = maximum permissible time 401 - Check the entered time Parameter value = maximum permissible time 600 - Check the entered value 700 - Check the value entered in speedLimitSom for axes, rpmLimitSom for spindles Parameter value = 2 -> SLS_2, Parameter value = 3 -> SLS_3, Parameter value = 4 -> SLS_4 800 - Check the entered additional functions 810 - Check the entered time 820 - Check the entered time 900 - Check the input 901 - Check the input 1000 - Check the input (maximum = 30 ms) 1100 - Check the input 1200 - Check the drive assignment - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 200 - Machine parameter axisGroup: Assigned axis group out of range **Error correction** - Check the ERR ID: 200 - Check the entered value of the axis group - Inform your service agency
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 201 - Machine parameter axisGroup: Entered axis group not in use. - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 201 - Enter only axis groups that are actually being used - Inform your service agency.

Error number	Description
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 202 - Drive type and axis group type do not match. - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 202 - Check the axis group assignment - Inform your service agency
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 210 - Permissible number of spindles was exceeded - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 210 - Check the number of spindles in your system - Inform your service agency
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 300 - Machine parameter pwmSignalOutput. Input range exceeded - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 300 - Check the machine parameter pwmSignalOutput. Input value too large. - Inform your service agency.

Error number	Description
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 301 - Machine parameter pwmSignalOutput. The same value was used in two separate entries - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 301 - Check the machine parameter pwmSignalOutput. The same value is entered in two separate entries. - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 400 - Machine parameter timeLimitStop1: Input range exceeded - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 400 - Check the entered time Parameter value = maximum permissible time - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 401 - Machine parameter timeLimitStop2: Input range exceeded - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 401 - Check the entered time Parameter value = maximum permissible time - Inform your service agency.

Error number	Description
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 600 - Machine parameter distPerMotorTurn: Input range exceeded - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 600 - Check the entered value - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 700 - Machine parameter speedLimitSom2, speedLimitSom3, speedLimitSom4, rpmLimitSom2, rpmLimitSom3, rpmLimitSom4: Permissible velocity limit exceeded - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 700 - Check the value entered in speedLimitSom for axes, rpmLimitSom for spindles Parameter value = 2 -> SLS_2, Parameter value = 3 -> SLS_3, Parameter value = 4 -> SLS_4 - Inform your service agency.

Error number	Description
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 800 - An unknown additional function is activated in the config object CfgSafety - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 800 - Check the entered additional functions - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 810 - Machine parameter timeToEmStopTest: Input range exceeded - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 810 - Check the entered time - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 820 - Machine parameter watchdogTime: Input range exceeded - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 820 - Check the entered time - Inform your service agency.

Error number	Description
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 900 - Machine parameter brakeAfter: Entered connective operation is not allowed - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 900 - Check the input - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 901 - Machine parameter brakeAfter: A connective operation to yourself is not allowed - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 901 - Check the input - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 1000 - Machine parameters plcCount: Input range PLC / SPLC-MC configured Cycle time was exceeded - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 1000 - Check the input (maximum = 30 ms) - Inform your service agency.

Error number	Description
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 1100 - Machine parameters idleState: Configuration of subsequent condition after a stop reaction SS2 is out of the permissible range - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 1100 - Check the input - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 1200 - The drive assignment is missing for a safe axis - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 1200 - Check the drive assignment - Inform your service agency.
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data for SKERN are faulty. The ERR ID provides information on the faulty machine parameter: 1201 - An inactive axis is marked as active in the functional safety environment (FS). - Faulty parameter values were entered - An internal software error has occurred **Error correction** - Check the ERR ID: 1201 - Check the axis configuration - Inform your service agency.

Error number	Description
231-E01A	**Error message** E01A CC%2 FS config error SS2 reaction ObjId=%5 Param=%6 **Cause of error** The permissible value range for the safe machine parameters distLimitStop2 or rpmLimitSom3AtSS2 was exceeded. ObjID = Number of the drive with incorrect configuration Param = Transferred value **Error correction** - Check the entered value - Inform your service agency
231-E01A	**Error message** E01A CC%2 FS config error SS2 reaction ObjId=%5 Param=%6 **Cause of error** The permissible value range for the safe machine parameters distLimitStop2 or rpmLimitSom2AtSS2 was exceeded. ObjID = Number of the drive with incorrect configuration Param = Transferred value **Error correction** - Check the entered value - Inform your service agency
231-E01A	**Error message** E01A CC%2 FS config error SS2 reaction ObjId=%5 Param=%6 **Cause of error** The permissible value range for safe machine parameters distLimitStop2 or rpmLimitSom4AtSS2 was exceeded. ObjID = Number of the drive with incorrect configuration Param = Transferred value **Error correction** - Check the entered value - Inform your service agency

Error number	Description
231-E01A	**Error message** E01A CC%2 FS configuration error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The transferred configuration data (safe machine parameters) for SKERN contain faulty data. The ERR-ID describes the error more precisely: 204 – Axis configured as safe axis (i.e. <axisGroup> unequal -1) But: The safety-relevant information ("safety bits") is not supported by the speed encoder although the EnDat2.2 mode of the encoder is active and it is a single-encoder system. This configuration is not permitted for a safe axis. CC: Number of the CC ObjId: Drive number (0 <=> X51, 1 <=> X52 etc.) Param: Axis group (value of the affected MP <axisGroup>) **Error correction** 204 - Check the speed encoder. It may have to be exchanged. – Check the axis configuration – Inform your service agency

Error number	Description
231-E01B	**Error message** E01B CC%2 SPLC program error ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The SPLC program has found an illegal value in an API marker or in an API word in the ApiToSafety (symbolic memory interface of SPLC to the safety software SKERN). The additional information provides exact information: ERR-ID = Information ObjId = Depends on the ERR-ID Param = Depends on the ERR-ID ERR_ID: 100: PP_AxGrpStateReq - Range violation SPLC-CC ObjektID = Axis group Parameter = Value of the required safety function 200: PP_AxGrpActivate - Illegal marker value ObjektID = Axis group Parameter = 1 - SPLC CC marker value illegal = 2 - SPLC MC marker value illegal 300: PP_AxFeedEnable - Illegal marker value ObjektID = Axis Parameter = 1 - SPLC CC marker value illegal = 2 - SPLC MC marker value illegal 400: PP_AxGrpStopReq - Range violation SPLC CC ObjektID = Axis group Parameter = Value of the required safety function 500: PP_AxGrpPB - Illegal marker value ObjektID = Axis group Parameter = 1 - SPLC CC marker value illegal = 2 - SPLC MC marker value illegal 600: PP_GenFB_NCC - Illegal marker value SPLC CC ObjektID = No meaning Parameter = No meaning 700: PP_GenCVO - Illegal marker value ObjektID = No meaning Parameter = 1 - SPLC CC marker value illegal = 2 - SPLC MC marker value illegal 800: PP_AxGrpPermitDrvOn - Illegal marker value ObjektID = Axis group Parameter = 1 - SPLC CC marker value illegal = 2 - SPLC MC marker value illegal 900: PP_GenMKG - Illegal marker value SPLC CC ObjektID = No meaning Parameter = No meaning 1000: PP_GenTBRK - Illegal marker value SPLC CC ObjektID = No meaning Parameter = No meaning **Error correction** - To set API words, use only the definitions of the corresponding datum in "SPlcApiMarker.def" - Check the range limits of the API word. - Check the values assigned to the API marker. - Inform your service agency.

Error number	Description
231-E01C	**Error message** E01C CC%2 Illegal FS function ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The SPLC program has requested an illegal safety function over PP_AxGrpStateReq in the ApiToSafety (symbolic memory interface of SPLC to the core safety software SKERN). ERR-ID = Exact information ObjId = Axis group causing the error Param = Depends on the ERR-ID ERR_ID: 100: Requested safety function is not supported Param = Requested safety function 200: The safety function SLI_S was requested for a axis group that is not of the spindle type. Param = No meaning 300: The safety function SLI_2, SLI_3 or SLI_4 was requested for an axis group that is of the spindle type. This safety function is not allowed for spindles. Param = Requested safety function 400: The safety function SLS_4 or SLI_4 was requested for a axis group and the function is not enabled. Param = Requested safety function 401: A direct change is requested from the safety function SLS_2, SLI_2, SLS_3 or SLI_3 into the safety function SLS_4 or SLI_4. This direct change is not allowed. Parameter = Requested safety function 402: A direct change is requested from the safety function SLS_4 or SLI_4 into the safety function SLS_2, SLI_2, SLS_3 or SLI_3. This direct change is not allowed. Parameter = Requested safety function 500: The SPLC requested the safety function STO or STO_O for an axis group whose drives are still in the control loop. The requested safety function can be requested only if all drives of this axis group have been switched off. Parameter = Value of the required safety function **Error correction** - Check the ERR ID: 100 - Use only safety functions that are supported with this software level. 200 - Request the safety function SLI_S only for axis groups with spindles. 300 - For spindles, request only the SLI_S function. 400 - Do not request a safety function or enable the safety related operating mode SOM 4 in machine parameter permitSom4. (Check the preconditions for enabling) 401 - Switch off the drives between the change of safety function and request SOS for at least one cycle. 402 - See 401

Error number	Description
	500 - Switch-off with the stop functions over PP_AxGrpStopReq. - Inform your service agency
231-E01C	**Error message** E01C CC%2 Illegal FS function ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The SPLC program has requested an illegal safety function over PP_AxGrpStateReq in the ApiToSafety (symbolic memory interface of SPLC to the core safety software SKERN). ERR-ID = Exact information ObjId = Axis group causing the error Param = Depends on the ERR-ID ERR_ID: 100: Requested safety function is not supported Param = Requested safety function **Error correction** - Check the ERR ID: 100 - Use only safety functions that are supported with this software level. - Inform your service agency
231-E01C	**Error message** E01C CC%2 Illegal FS function ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The SPLC program has requested an illegal safety function over PP_AxGrpStateReq in the ApiToSafety (symbolic memory interface of SPLC to the core safety software SKERN). ERR-ID = Exact information ObjId = Axis group causing the error Param = Depends on the ERR-ID ERR_ID: 200: The safety function SLI_S was requested for a axis group that is not of the spindle type. Param = No meaning **Error correction** - Check the ERR ID: 200 - Request the safety function SLI_S only for axis groups with spindles. - Inform your service agency

Error number	Description
231-E01C	**Error message** E01C CC%2 Illegal FS function ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The SPLC program has requested an illegal safety function over PP_AxGrpStateReq in the ApiToSafety (symbolic memory interface of SPLC to the core safety software SKERN). ERR-ID = Exact information ObjId = Axis group causing the error Param = Depends on the ERR-ID ERR_ID: 300: The safety function SLI_2, SLI_3 or SLI_4 was requested for a axis group that is not of the spindle type. This safety function is not permissible for spindles. Param = Requested safety function **Error correction** - Check the ERR ID: 300 - For spindles, request only the SLI_S function. - Inform your service agency
231-E01C	**Error message** E01C CC%2 Illegal FS function ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The SPLC program has requested an illegal safety function over PP_AxGrpStateReq in the ApiToSafety (symbolic memory interface of SPLC to the core safety software SKERN). ERR-ID = Exact information ObjId = Axis group causing the error Param = Depends on the ERR-ID ERR_ID: 400: The safety function SLS_4 or SLI_4 was requested for a axis group and the function is not enabled. Param = Requested safety function **Error correction** - Check the ERR ID: 400 - Do not request the safety function, or enable Bit #0 in MP560 Bit#0. (Check the preconditions for enabling) - Inform your service agency

Error number	Description
231-E01C	**Error message** E01C CC%2 Illegal FS function ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The SPLC program has requested an illegal safety function over PP_AxGrpStateReq in the ApiToSafety (symbolic memory interface of SPLC to the core safety software SKERN). ERR-ID = Exact information ObjId = Axis group causing the error Param = Depends on the ERR-ID ERR_ID: 401: A direct change is requested from the safety function SLS_2, SLI_2, SLS_3 or SLI_3 into the safety function SLS_4 or SLI_4. This direct change is not allowed. Parameter = Requested safety function **Error correction** - Check the ERR ID: 401 - Switch off the drives between the change of safety function and request SOS for at least one cycle. - Inform your service agency
231-E01C	**Error message** E01C CC%2 Illegal FS function ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The SPLC program has requested an illegal safety function over PP_AxGrpStateReq in the ApiToSafety (symbolic memory interface of SPLC to the core safety software SKERN). ERR-ID = Exact information ObjId = Axis group causing the error Param = Depends on the ERR-ID ERR_ID: 402: A direct change is requested from the safety function SLS_4 or SLI_4 into the safety function SLS_2, SLI_2, SLS_3 or SLI_3. This direct change is not allowed. Parameter = Requested safety function **Error correction** - Check the ERR ID: 401 - Switch off the drives between the change of safety function and request SOS for at least one cycle. 402 - See 401 - Inform your service agency

Error number	Description
231-E01C	**Error message** E01C CC%2 Illegal FS function ERR-ID=%4 ObjId=%5 Param=%6 **Cause of error** - The SPLC program has requested an illegal safety function over PP_AxGrpStateReq in the ApiToSafety (symbolic memory interface of SPLC to the core safety software SKERN). ERR-ID = Exact information ObjId = Axis group causing the error Param = Depends on the ERR-ID ERR_ID: 500: The SPLC requested the safety function STO or STO_O for an axis group whose drives are still in the control loop. The requested safety function can be requested only if all drives of this axis group have been switched off. Parameter = Value of the required safety function **Error correction** - Check the ERR ID: 500 - Switch-off with the stop functions over PP_AxGrpStopReq. - Inform your service agency
231-E01D	**Error message** E01D CC%2 SKERN Start requirement not fulfilled Id=%4, V=%4 **Cause of error** - The requirements for starting cyclic operation are not fulfilled. **Error correction** - ID=1: Configuration was not successful. Check the additional information of the error messages E01A or E019. - ID=2: Check whether an error message E01E occurred. - ID=3: Your hardware version does not fulfill the requirements for safe operation. - Inform your service agency
231-E01E	**Error message** E01E CC%2 SKERN Version comparison failed **Cause of error** - In the SKERN software version comparison, a difference between the MC and CC was found. **Error correction** - Check the MC and CC software versions and make sure to use identical software versions on CC and MC. - Inform your service agency

Error number	Description
231-E01F	**Error message** E01F CC%2 SPLC program requests invalid change of operating mode **Cause of error** - The SPLC program requests an illegal change of the safe operating mode via the date ApiToSafety.PP_GenSOM. **Error correction** - Check the SPLC program. The following changes are not permitted: S_MODE_SOM_2 -> S_MODE_SOM_4 S_MODE_SOM_3 -> S_MODE_SOM_4 S_MODE_SOM_4 -> S_MODE_SOM_2 S_MODE_SOM_4 -> S_MODE_SOM_3 - Inform your service agency
231-E020	**Error message** E020 CC%2 Incorrect SPLC-RTS data ID=%4 Info1=%5 Info2=%6 **Cause of error** - When the cyclic data from the SPLC were checked, an error was found. **Error correction** - Check the software versions - Inform your service agency

Error number	Description
231-E021	**Error message** E021 Cut-out signal status incorrect CC%2 Nom:%4 ErrMask: %5 Sgn:%6 **Cause of error** An error occurred in the self-test for safety. A faulty output condition was found before the cut-out channel test: At least on of the examined cut-out signals has the wrong condition. These are the meanings of the abbreviations: Nom (nominal condition of the cut-out signals, decimal notation, interpret in binary): 0: Lock / 1: Release through the corresponding cut-out signal (bits 0 to 9 of Nom) Bit0:STOS.A.MC, Bit1:STO.A.MC.WD, Bit2:STO.A.P.x, Bit3:S-TO.A.PIC, Bit4:STO.B.CC.WD Bit5:STO.B.P.x, Bit6:STO.A.CC, Bit7:STO.A.SPL.WD, Bit8:S-TOx.A.RES, Bit9:STO.A.T ErrMask (error mask, decimal notation, interpret in binary): The corresponding switch-off signal (bits 0 – 9 of ErrMask) locks although it should release, or releases although it should lock. Sgn (cut-out signal to be tested): 0: STOS.A.MC 1: STO.A.MC.WD 2: STO.A.P.x 3: STO.A.PIC 4: STO.B.CC.WD 5: STO.B.P.x 6: STO.A.CC 7: STO.A.SPL.WD 8: STOx.A.RES 9: STO.A.T **Error correction** - Check the wiring (PWM cable) - Replace the defective hardware (power module, controller unit) - Inform your service agency
231-E022	**Error message** E022 CC%2 SKERN-CC: Error in configuration data ID=%4 **Cause of error** A data error was found during cyclic checking of the configuration data. - Internal software error **Error correction** - Create a service file - Inform your service agency

Error number	Description
231-E023	**Error message** E023 CC%2 dynamic test ID=%4 Info1=%5 Info2=%6 **Cause of error** - An error was found during the dynamic test (minute test) **Error correction** - Inform your service agency.
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Time exceeded in A channel test **Cause of error** The dynamic test of the PLC peripheral devices (PL, MB, UEC) was cancelled during the A-channel test. - Internal software error - Error in the HSCI communication between MC and HSCI peripheral device **Error correction** - Exchange the defective HSCI peripheral component. - Check the software version. - Inform your service agency.
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: CC software interrupt mask %5 **Cause of error** The CC interrupt mask was changed during the dynamic test of the B-channel peripheral devices (PL, MB, UEC). - Internal software error - Defective peripheral device - Defective CC 61xx controller unit **Error correction** - Check the software version. - Exchange the defective CC 61xx - Exchange the defective peripheral device - Inform your service agency.

Error number	Description
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Invalid test ID %5 **Cause of error** During the dynamic test of the PLC peripheral devices (PL, MB, UEC) of the B channel, an incorrect test ID was transferred from the MC to the CC. It is possible that the connected periphery does not match the configured periphery. - Incorrect peripheral device configured - Internal software error - Disturbance of the HSCI communication **Error correction** - Use the HSCI-BUS diagnostics to check the connected periphery. - If required, correct the HSCI configuration. - Check the software version. - Exchange the peripheral device. - Inform your service agency.
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Error in B channel PL input no. %5 **Cause of error** The dynamic test of the PLC peripheral devices (PL, MB, UEC) discovered that an input to be tested did not switch to the 0 level. - Incorrect S machine parameters SMP587 - Wiring error of the test groups/test outputs - Error in the configuration file (IOC file) - Disturbance of HSCI communication **Error correction** - Check the entry in the machine parameter SMP587 - Check the HSCI-BUS diagnostics - Check the wiring and HSCI configuration - Exchange the peripheral device. - Inform your service agency.

Error number	Description
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: -ES.B inactive test ID=%5 **Cause of error** The dynamic test of the PLC peripheral devices (PL, MB) discovered that an ES.B input was not activated or the input is defective. - Wiring error of the test groups/test outputs - Error in the configuration file (IOC file) **Error correction** - Check the HSCI-BUS diagnostics. If necessary, correct the HSCI configuration - Check/correct the wiring - Exchange the peripheral device. - Inform your service agency.
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: -ES.B.HW inactive test ID=%5 **Cause of error** The dynamic test of the PLC peripheral devices (MB, PL) discovered that an ES.B.HW input was not activated or the input is defective. - Wiring error of the test groups/test outputs - Error in the HSCI configuration file (IOC file) **Error correction** - Check the HSCI-BUS diagnostics. If necessary, correct the HSCI configuration - Check/correct the wiring - Exchange the peripheral device. - Inform your service agency.
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Error in B channel: test ID=%5 **Cause of error** The dynamic test of the PLC peripheral devices (PL, MB, UEC) discovered an error in the test flow. An incorrect ID appeared at the end of the test. - Internal software error **Error correction** - Check the software version. - Inform your service agency.

Error number	Description
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Error in B channel: test ID=%5 **Cause of error** The dynamic test of the PLC peripheral devices (PL, MB, UEC) was was not conducted for the given test ID. - Error in the configuration file (IOC file) - Internal software error **Error correction** - Check the HSCI-BUS diagnostics. If necessary, correct the HSCI configuration - Check the software version. - Inform your service agency.
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Processor check error %5 **Cause of error** Error in the flow of the dynamic test - Internal software error **Error correction** - Check the software version. - Inform your service agency.
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Error in B channel: test ID=%5 **Cause of error** The dynamic test of the PLC peripheral devices (PL, MB, UEC) was was not conducted for the given test ID. - Error in the HSCI configuration file (IOC file) - Internal software error **Error correction** - Check the HSCI-BUS diagnostics. If necessary, correct the HSCI configuration - Check the software version. - Inform your service agency.

Error number	Description
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Error in B channel: test ID=%5 **Cause of error** Error during the dynamic test of the PLC periphery devices (PL, MB, UEC) Missing acknowledgment of the TEST.B output. - Incorrect wiring of the TEST.B output. - Short circuit of the TEST.B output - Incorrect firmware version of the peripheral device **Error correction** - Check the software version. - Check the wiring of the TEST.B output - Run a software or firmware update - Exchange the faulty device. - Inform your service agency.
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Error in B channel: test ID=%5 **Cause of error** Error during the dynamic test of the PLC periphery devices (PL, MB, UEC). No expected switch from 1 to 0 of the test output TEST.B detected. - Incorrect wiring of the TEST.B output. - Short circuit of the TEST.B output - Incorrect firmware version of the peripheral device **Error correction** - Check the software version. - Check the wiring of the TEST.B output - Run a software or firmware update - Exchange the faulty device. - Inform your service agency.

Error number	Description
231-E023	**Error message** E023 CC%2 Dynamic test ID=%4 aborted: Error in B channel: test ID=%5 **Cause of error** Error during the dynamic test of the PLC periphery devices (PL, MB, UEC). No expected switch from 0 to 1 of the test output TEST.B detected. - Incorrect wiring of the TEST.B output. - Short circuit of the TEST.B output - Incorrect firmware version of the peripheral device **Error correction** - Check the software version. - Check the wiring of the TEST.B output - Run a software or firmware update - Exchange the faulty device. - Inform your service agency.
231-E024	**Error message** E024 CC%2 alarm self-test Addr:%4, Dev:%5, Actl:%6, Noml: %7 **Cause of error** The self-test detected an internal process error. **Error correction** - Inform your service agency.
231-E025	**Error message** E025 CC%2 error: self-test ErrId:%4, Par1:%5, Par2:%6, Par3: %7 **Cause of error** The self-test detected an internal process error. **Error correction** - Check the software versions - Inform your service agency.
231-E026	**Error message** E026 CC%2 Illegal MC software installed **Cause of error** An illegal software version of the MC software (autotest version) was detected. **Error correction** - Check the software versions - Inform your service agency.

Error number	Description
231-E027	**Error message** E027 CC%2 Error: Safety function STO AxGrp=%3 ErrId=%4 **Cause of error** The Safe Torque Off safety function (STO) was not complied with for the given axis group. **Error correction** - Check whether this is an aftereffect and, if so, correct its cause. - Inform your service agency.
231-E028	**Error message** E028 CC%2 MB/PLB does not respond. Device ID: %3 **Cause of error** - HSCI components with functional safety (FS) no longer responds (B channel): MB machine operating panel or PLB 6xxx - Component was separated during operation with the HSCI bus - 24 V NC power supply of the component was interrupted - The entered device ID provides information on the affected parameter: 5 = System module on PLB 62xx FS 7 = MB 6xx FS 15 = Expansion PLB (without system module) PLB 61xx FS, PLB 60xx FS 17 = Integrated PLB of the UEC controller unit - Faulty firmware of the PLB or the MB **Error correction** - Check the MB and PLB in the HSCI bus diagnosis - Save service files - Check the HSCI connections - Check the 24 V power supply of the MB and the PLB - Inform your service agency

Error number	Description
231-E029	**Error message** E029 CC%2 Test step not possible (ID=%3) **Cause of error** One of the following steps in the safety self-test is not possible in a drive system with DRIVE-CLiQ inverters due to the hardware. The safety self-test can be configured through safe machine parameters in the configuration datum with CfgSafety. **Error correction** The given ID number indicates the appropriate test that must be deactivated for a system with DRIVE-CLiQ inverters. ID = 100: Test of the brake control. Test the setting in machine parameter testNotBrakeLine. ID = 200: Cutout channel test via signals STO.A.G / STOS.A.G. Check the setting in machine parameter testNotStoGlobal. ID = 300: Cutout channel test via signals internal to the control. Test the setting in machine parameter testNotStoIntrnl. - Note: These settings must be changed only by the machine tool builder. - Inform your service agency.
231-E02A	**Error message** E02A CC%2 SKERN-CC: Response: norm. closed contact unequal A=%4, B=%5 **Cause of error** The acknowledgment signal of the normally closed contact chain PP_GenFB_NCC of the A channel does not match the signal of the B channel. **Error correction** - Check signals FB_NCC.A and FB_NCC.B - Check the wiring of the normally closed contact chain - Check the SPLC program - Inform your service agency
231-E030	**Error message** E030 CC%2 process error in safety self-test %4 **Cause of error** An internal process error has occurred during the self test: - Internal software error **Error correction** - Save the service file - Restart the control. - Inform your service agency.

Error number	Description
231-E031	**Error message** E031 CC%2 FS outputs not "0" 0-31:%4 32-63:%5 E033 CC%2 FS outputs not "0" 64-95:%4 96-127:%5 **Cause of error** - In the safety self-test, the WD.B.SPL FS outputs remain set during the watchdog test, although they were supposed to be reset. 0-31: Bit-encoded status of the FS outputs 0 to 31 31-63: Bit-encoded status of the FS outputs 32 to 63 64-95: Bit-encoded status of the FS outputs 64 to 95 96-127: Bit-encoded status of the FS outputs 96 to 127 - Short circuit of an FS output on +24 V - Hardware defective **Error correction** - Check the wiring of the FS outputs. - Exchange the hardware - Generate the service files and notify the Service Department
231-E032	**Error message** E032 CC%2 power supply unit not ready for operation **Cause of error** - In the safety self-test, the supply unit does not indicate readiness for operation - The RDY.PS signal is inactive for one of the following reasons: - No release at the X70 connector of the supply unit - Short circuit of the KDR commutation reactor - Incorrect wiring of the KDR - Hardware defective **Error correction** - Check the wiring on the connector X70 - Check the KDR wiring - Exchange the hardware - Generate the service files and notify the Service Department
231-E110	**Error message** E110 Timeout in current measurement %1 **Cause of error** - Current measuring during safety self-test exceeds the defined time **Error correction** - Internal software error - Inform your service agency

Error number	Description
231-E120	**Error message** E120 CC safe function call error **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version
231-E130	**Error message** E130 Test current is too small %1 **Cause of error** - The test current measured in the measurement of the current during the safety self-test is too small - The current control loop has not been adjusted - Drive enabling at PWM output is missing at the time of measurement of the current - The current sensor is defective **Error correction** - As a test, switch off the measurement of the current in CfgSafety via safe machine parameter testNotCurrent. The resulting error messages provide information on possibly faulty signals for drive enabling. - Adjust the current controller - Check the current sensor - Inform your service agency
231-E140	**Error message** E140 Mot. current %1 not equal 0 **Cause of error** - The current measured in the pulse deletion test is too great - One of the cutout channels "-AP1.x", "-SH1AB", "-AP2.x" or "-SH2.WD" is not effective **Error correction** - Check the wiring - Inform your service agency
231-E150	**Error message** E150 RDY.x status stays active %1 **Cause of error** - The measured status of the 'RDY.x' signal is active - The 'RDY.x' signal does not change to the 'inactive' ('low' level) status, although the MC is blocking the power module through a corresponding enabling signal. **Error correction** - Internal software error - Hardware defective - Inform your service agency

Error number	Description
231-E160	**Error message** E160 RDY.x status is inactive %1 **Cause of error** - The measured status of the 'RDY.x' signal is inactive - The 'RDY.x' signal does not change to the 'active' ('high' level) status, although the MC is blocking the power module through a corresponding enabling signal. **Error correction** - Internal software error - Hardware defective - Inform your service agency
231-E170	**Error message** E170 Pos. error too large %1 **Cause of error** - Defect in the mounting of the position encoder - Incorrect thermal, linear or nonlinear compensation - Excessive backlash **Error correction** - Check the parameter value (maximum position deviation between MC and CC during operation) - Check the parameter value in CfgAxisComp->linearComp-Value (linear axis error compensation for analog axes) - Check the parameter value in CfgAxis-Comp->backLashType1 (backlash compensation) - Check the mounting of the position encoder- Inform your service agency
231-E180	**Error message** E180 Z1-track amplitude too high %1 **Cause of error** - The amplitude of the Z1-track encoder signal for the speed encoder is too high - Noise on motor encoder signal - Short circuit in the motor encoder cable - Z1-track signal amplitude of motor encoder too high **Error correction** - Check the connection of the motor encoder - Check the cable of the speed encoder - Check the motor encoder - Inform your service agency

Error number	Description
231-E190	**Error message** E190 RDY.x status of axes stays active (safety relay) %1 **Cause of error** - Measured status of the '-STO.A.G' signals remain inactive during the test (high level) - The '-STO.A.G' signal does not change to the 'active' status (low level), even though the MC sets the corresponding signal status - The power stage (inverter) of at least 1 feed axis or auxiliary axis is not locked through the '-STO.A.G' signal. **Error correction** - Check the wiring - Hardware defective - Inform your service agency
231-E1A0	**Error message** E1A0 RDY.x status of axes stays inactive (safety relay) %1 **Cause of error** - Measured status of the '-STO.A.G' signals remain active during the test (high level) - The '-STO.A.G' signal does not change to the 'inactive' status (high level), even though the MC sets the corresponding signal status - The power stage (inverter) of at least 1 feed axis or auxiliary axis is locked through the '-STO.A.G' signal. **Error correction** - Check the wiring - Hardware defective - Inform your service agency
231-E1C0	**Error message** E1C0 RDY.x status of spindle stays active (safety relay) %1 **Cause of error** - Status of the '-STOS.A.G' signals remain inactive during the test (high level) - '-STOS.A.G' signal does not change to the 'active' status (low level), even though the MC assigns the corresponding signal status - The power stage (inverter) of the spindle is not locked through the '-STOS.A.G' signal. **Error correction** - Check the wiring - Hardware defective - Inform your service agency

Error number	Description
231-E1E0	**Error message** E1E0 CC%2 RDY.x status remains active (safety relay)%1 **Cause of error** - An error occurred during the watchdog test WD.A.STO of the PL 6xxxFS. A power stage (inverter) is not switched off through the -STOS.A.G or -STO.A.G signal. - Faulty or missing wiring of -STO.A.G, -STOS.A.G or X71, X72 of the power module (UV, UE) - Error in parameters of the control of X71, X72 through -STO.A.G, -STOS.A.G in SMP - Hardware defective **Error correction** - Check the wiring - Check SMP - Exchange the hardware - Generate the service files and notify the Service Department
231-E200	**Error message** E200 Timeout during emergency stop (SS1) %1 **Cause of error** - The maximum permissible braking time for stopping at the emergency braking ramp (SS1 reaction) was exceeded **Error correction** - Check the machine parameter: timeLimitStop1: Default time for stopping at the emergency braking ramp for SS1 reaction - Inform your service agency

Error number	Description
231-E220	**Error message** E220 Standstill monitoring SKERN-CC %1 **Cause of error** - SKERN-CC detected an impermissibly large axis movement in the SOS safety condition. The standstill speed, however, was not exceeded. The maximum permissible path is defined in the machine parameter MP positionRangeVmin . Possible causes: - Machine parameter "positionRangeVmin" is defined too small. - The brake was deactivated before the position controller was closed. - The brake was not activated before the position controller was opened. - When an axis was switched on, some existing following error was corrected. - The brake is defective. - There was an attempt to move an axis in the SOS condition (PLC?) **Error correction** - Check the entry in machine parameter positionRangeVmin. - Check the sequence of deactivating the brake and closing the position controller. - Check the sequence of activating the brake and opening the position controller. - Check whether there is a following error after an axis is locked. - Inform your service agency.
231-E230	**Error message** E230 Axis %1 STO safety function not fulfilled **Cause of error** The Safe Torque Off safety function (STO) was not complied with for the given axis. **Error correction** - Check whether this is an aftereffect and, if so, correct its cause. - Inform your service agency.

Error number	Description
231-E240	**Error message** E240 Axis %1 faulty braking process **Cause of error** - Axis was incorrectly braked during an SS1 reaction. Possible causes: - Adjusted delay time of the dv/dt monitoring (timeToleranceDvDt) is insufficient. An acceleration might occur after expiration of the delay time. - Drive is not optimally adjusted - Brake ramp in machine parameter motEmergencyStopRamp is not correctly adjusted **Error correction** - Check the entry in machine parameter timeToleranceDvDt - Check the adjusted braking ramp for emergency stops in machine parameter motEmergencyStopRamp - Check the velocity of the drive during the braking process with the control's internal oscilloscope - Adjust the drive. - Inform your service agency

Error number	Description
231-E250	**Error message** E250 Error in cut-out channel %1 Ch:%4 St:%5 St-2ndCh:%6 Sgn:%7 **Cause of error** An error occurred in the self-test for safety. The cut-out channel test detected a fault. Key to abbreviations: Ch (cut-out channel concerned): 1: STO.A.x 2: STO.B.x 3: STO.A.G 4: STOS.A.G St (actual condition of the cut-out channel): 0: Cut-out channel is inactive, although it is supposed to be active 1: Cut-out channel is active, although it is supposed to be inactive St-2ndCh (actual condition of the second cut-out channel): 0: Cut-out channel is inactive 1: Cut-out channel is active The second cutout channel is STO.A.x if Ch=2, and STO.B.x if Ch=1, Ch=3 and Ch=4 Sgn (cut-out signal concerned): 0: STOS.A.MC 1: STO.A.MC.WD 2: STO.A.P.x 3: STO.A.PIC 4: STO.B.CC.WD 5: STO.B.P.x 9: STO.A.T 99: no cut-out signal **Error correction** - Check the wiring (PWM cable) - Replace the defective hardware (power module, controller unit) - Inform your service agency.
231-F000	**Error message** F000 CC%2 S-function call error %1 **Cause of error** - Internal software error **Error correction** - Inform your service agency - Check the software version

Error number	Description
231-F100	**Error message** F100 No brake test was conducted %1 **Cause of error** - MC runs no test of the motor brake(s) although it is necessary according to parameter settings. - The call for testing a motor brake lasts longer than 5 seconds. **Error correction** - Inform your service agency
231-F200	**Error message** F200 No brake line test was conducted %1 **Cause of error** - MC runs no test of the motor brake line although it is necessary according to parameter settings. - The call for testing a brake line lasts longer than 10 seconds. **Error correction** - Inform your service agency
231-F300	**Error message** F300 CC%2 Cancelation of the cutout channel test %1 **Cause of error** - The switch-off test was ended automatically because the maximum permissible delay time was exceeded. - The NC did not properly finish a test section. - The NC does not perform a certain test. **Error correction** - Check whether a previous system error of the NC led to the cancelation of the test section - Check the software version - Inform your service agency
234-0001	**Error message** Insufficient memory **Cause of error** Too many clients are configured for the Ethernet transmission. **Error correction** Inform your service agency
234-0002	**Error message** HSCI Ethernet connection interrupted **Cause of error** The Ethernet transmission is disturbed. **Error correction** - Check the cabling - Inform your service agency

Error number	Description
234-0003	**Error message** HSCI Ethernet configuration without CC **Cause of error** If the HSCI is configured or connected, a CC must also be connected to the HSCI. **Error correction** - Check the cabling - Inform your service agency
234-0004	**Error message** Internal software error **Cause of error** Not enough memory for the job at hand **Error correction** Inform your service agency
234-0005	**Error message** Internal software error **Cause of error** HSCI initialization not completed **Error correction** Inform your service agency
234-0006	**Error message** Internal software error **Cause of error** High-speed inputs not initialized on the PLB 6xxx (HSCI). **Error correction** Inform your service agency
234-0007	**Error message** Fast input does not exist **Cause of error** A high-speed input on a PLB 6xxx (HSCI) adjusted in the configuration data of the control is missing. **Error correction** - Ensure that the input has been configured with IOconfig. - Please keep in mind that the input has to be of the bit data type. - Dual-channel inputs (FS) cannot be used.

Error number	Description
234-0008	**Error message** HSCI total amount of data too great **Cause of error** The number of permissible HSCI data telegrams or their total size was exceeded. **Error correction** Inform your service agency.
234-0009	**Error message** Too many HSCI telegrams **Cause of error** The maximum number of HSCI datagrams was exceeded. The IO configuration contains too many HSCI devices. **Error correction** Inform your service agency.
234-000A	**Error message** HSCI data has reached critical size. **Cause of error** The total number of HSCI data has reached a critical size. This increases the risk of transmission errors. **Error correction** - Reduce the number of HSCI components on the HSCI bus. - Inform your service agency.
234-000B	**Error message** HSCI data size too large for HSCI device **Cause of error** The permissible total data amount for an HSCI device has been exceeded. **Error correction** Inform your service agency.
234-000C	**Error message** Impermissible HSCI device detected on X501 of the MC **Cause of error** At least one HSCI device was detected on the X501 connection of the MC that is not suited for operation on this connection. **Error correction** Check the devices connected to X501. The following devices are not allowed on this connection: - CC controller units (including UECs and UMCs) - PL 6xxx FS - More than one dual-channel machine operating panel (e.g. MB 620FS, PL 6001FS)

Error number	Description
234-000D	**Error message** Error during initialization of the SPI module (MCU) **Cause of error** Could not create memory for the SPI inputs and outputs. **Error correction** Switch the control off and back on. If the error recurs, inform your service agency.
234-000E	**Error message** Maximum number of controller units exceeded **Cause of error** There are too many CC controller units connected to the HSCI bus. **Error correction** - Note the maximum number of the respective HSCI participants. Refer to the Technical Manual for your control for more information about this. - Check the HSCI configuration. - Inform your service agency.
234-000F	**Error message** Maximum number of PLB or MB exceeded **Cause of error** There are too many PLB 6xxx units or MB machine operating panels connected to the HSCI bus. **Error correction** - Note the maximum number of the respective HSCI participants. Refer to the Technical Manual for your control for more information about this. - Check the HSCI configuration. - Inform your service agency.
234-0010	**Error message** Maximum number of HSCI devices exceeded **Cause of error** Too many HSCI devices (CC + UxC + PL 6xxx + MB) were detected on the HSCI bus. **Error correction** - Note the maximum number of the respective HSCI participants. Refer to the Technical Manual for your control for more information about this. - Check the HSCI configuration. - Inform your service agency.

Error number	Description
234-0011	**Error message** HSCI handwheel configuration error **Cause of error** Too many handwheels were detected on HSCI devices, or the handwheels are not allowed for this control **Error correction** - Check the configuration and number of connected handwheels - Inform your service agency
234-0012	**Error message** HSCI touch probe configuration error **Cause of error** Too many touch probes were detected on HSCI devices, or the touch probes are not allowed for this control **Error correction** - Check the configuration and number of touch probes - Inform your service agency
234-0013	**Error message** Error while reading the HSCI information **Cause of error** An error occurred during the import of hardware information for HSCI initialization **Error correction** - Restart the control - Check the HSCI cabling - Inform your service agency
234-0014	**Error message** Fault an HSCI interface X500 of the MC **Cause of error** - Problems at the HSCI interface X500 of the MC main computer - Bad or missing HSCI connection with X500 **Error correction** - Check the connecting element at X500 of the MC - Check the HSCI cable and HSCI devices connected at X500 - Defective main computer (MC) - Inform your service agency

Error number	Description
234-0015	**Error message** Fault an HSCI interface X501 of the MC **Cause of error** - Problems at HSCI interface X501 of the main computer (MC) - Bad or missing HSCI connection with X501 **Error correction** - Check the connecting element at X501 of the MC - Check the HSCI cable and the HSCI devices connected at X501 - Defective main computer (MC) - Inform your service agency
234-0017	**Error message** HSCI data not updated **Cause of error** - HSCI data have not been updated since the last cycle - Internal software error **Error correction** - Check the HSCI connections and the power supply for HSCI devices If the error recurs: - Create service files and save them - Inform your service agency
234-0019	**Error message** Incorrect firmware version of the HSCI master component **Cause of error** The firmware version of the HSCI master FPGAs is incompatible with the installed NC software version. **Error correction** Inform your service agency.
234-001A	**Error message** HSCI telegram list too long **Cause of error** - Too many HSCI devices are connected or configured on the HSCI bus of the main computer (MC). - Main computer (MC) is defective **Error correction** - Reduce the number of devices in the HSCI configuration or HSCI bus - If the error occurs without previous change of the HSCI configuration, then the MC has a hardware error. In this case, exchange the main computer (MC). - Inform your service agency

Error number	Description
234-001C	**Error message** HSCI communication interrupted **Cause of error** During operation, an illegal change in the number of HSCI participants was found on the HSCI bus. You can find additional information on the error location in the bus diagnostics of the control or through the INTERNAL INFO soft key. **Error correction** - Check the HSCI cable connections and HSCI devices including their power supply. - Inform your service agency.
234-001D	**Error message** Error during HSCI initialization **Cause of error** Necessary memory areas could not be created during initialization of the HSCI master component. **Error correction** - Restart the control If the error recurs: - Create service files and save them - Inform your service agency
234-001E	**Error message** Initialization error of HSCI interface X500 **Cause of error** An error occurred during initialization of the interface component for the HSCI interface X500 **Error correction** - Restart the control If the error recurs: - Create service files and save them - Inform your service agency
234-001F	**Error message** Initialization error of HSCI interface X501 **Cause of error** An error occurred during initialization of the interface component for the HSCI interface X501 of the main computer (MC) **Error correction** - Restart the control If the error recurs: - Create service files and save them - Inform your service agency

Error number	Description
234-0020	**Error message** HSCI communication error **Cause of error** The received frame number of an HSCI frame does not match the expected frame number. **Error correction** If the error recurs: - Create service files and save them - Inform your service agency
234-0021	**Error message** HSCI communication error **Cause of error** The DMA from the MC memory to the HSCI master is still active when the HSCI transfer starts. Possible causes: - Internal software error - High number of "failed frames" **Error correction** If the error occurs together with a large number of failed frames, check the HSCI connections. The number of failed frames is shown in the bus diagnostics of the MC HSCI master. If the error recurs: - Create service files and save them - Inform your service agency
234-0022	**Error message** HSCI communication error **Cause of error** There has been no DMA from the MC memory to the HSCI master since the last HSCI data exchange. Possible causes: - Internal software error - High number of "failed frames" **Error correction** If the error occurs together with a large number of failed frames, check the HSCI connections. The number of failed frames is shown in the bus diagnostics of the MC HSCI master. If the error recurs: - Create service files and save them - Inform your service agency

Error number	Description
234-0024	**Error message** HSCI communication error **Cause of error** The HSCI transfer jobs were not yet completed when a new DMA transfer began from the MC memory to the HSCI master. Possible causes: - Internal software error - Too many “failed frames” **Error correction** If the error occurs together with a large number of failed frames, check the HSCI connections. The number of failed frames is shown in the bus diagnostics of the MC HSCI master. If the error recurs: - Create service files and save them - Inform your service agency
234-0025	**Error message** HSCI communication error **Cause of error** - HSCI communication error (by DMA) has occurred. **Error correction** - Create service files and save them - Inform your service agency
234-0026	**Error message** HSCI communication error **Cause of error** - HSCI communication error has occurred. **Error correction** - Create service files and save them - Inform your service agency
235-0001	**Error message** Installation error **Cause of error** The firmware file could not be opened. It was either not found or is faulty. **Error correction** Inform your machine tool builder
235-0002	**Error message** Installation error **Cause of error** The firmware file could not be opened because it is faulty. **Error correction** Inform your service agency

Error number	Description
235-0003	**Error message** Hardware error **Cause of error** Error during download of the firmware file. Either an error occurred during data transmission of the file, or the file has a wrong firmware version. **Error correction** Inform your machine tool builder
235-0004	**Error message** Hardware error **Cause of error** Timeout when downloading the firmware file. The CCU did not respond within the expected time. CCU is defective or the firmware version is incorrect. **Error correction** Inform your machine tool builder
235-0005	**Error message** Hardware error **Cause of error** Timeout during the checksum calculation. The CCU did not respond within the expected time. CCU is defective or the firmware version is incorrect. **Error correction** Inform your machine tool builder
235-0006	**Error message** Hardware error **Cause of error** Firmware checksum check failed. Incorrect CCU checksum. CCU is defective or the firmware version is incorrect. **Error correction** Inform your machine tool builder
235-0007	**Error message** Hardware error **Cause of error** Timeout after downloading the first part of the firmware. The CCU did not respond within the expected time. CCU is defective or the firmware version is incorrect. **Error correction** Inform your machine tool builder

Error number	Description
235-0008	**Error message** Hardware error **Cause of error** Timeout after downloading the second part of the firmware. The CCU did not respond within the expected time. CCU is defective or the firmware version is incorrect. **Error correction** Inform your machine tool builder
235-0009	**Error message** Hardware error **Cause of error** Timeout when checking whether the bootcode is running. The CCU did not respond within the expected time. CCU is defective or the bootcode version is incorrect. **Error correction** Inform your machine tool builder
235-000A	**Error message** Installation error **Cause of error** The bootcode file could not be opened. It was either not found or is faulty. **Error correction** Inform your machine tool builder
235-000B	**Error message** Installation error **Cause of error** The bootcode file could not be read because it is faulty. **Error correction** Inform your service agency
235-000C	**Error message** Hardware error **Cause of error** There are too many HSCI devices connected to the control. **Error correction** Remove some of the HSCI devices. Contact your machine tool builder.

Error number	Description
235-000D	**Error message** Hardware error **Cause of error** HSCI device is not running. The CCU did not respond within the expected time. CCU is defective or the firmware version is incorrect. **Error correction** Inform your machine tool builder
235-000E	**Error message** Hardware error **Cause of error** This CCU is not supported. An unknown CCU is connected to the control. **Error correction** Inform your machine tool builder
235-000F	**Error message** Hardware error **Cause of error** Timeout when checking whether the CCU runs. The CCU did not respond within the expected time. CCU is defective or the firmware version is incorrect. **Error correction** Inform your machine tool builder
235-0010	**Error message** Installation error **Cause of error** Configuration settings for CCU422 missing **Error correction** Inform your service agency
235-0011	**Error message** Installation error **Cause of error** Configuration settings for CCU422 are faulty **Error correction** Inform your service agency
235-0012	**Error message** Hardware error **Cause of error** Error during asynchronous data transfer **Error correction** Inform your machine tool builder

Error number	Description
235-0013	**Error message** Hardware error **Cause of error** Timeout during identification of the CCU software. The CCU did not respond within the expected time. CCU is defective or the firmware version is incorrect. **Error correction** Inform your machine tool builder
235-0014	**Error message** Installation error **Cause of error** Invalid initialization parameters. The configuration settings are faulty. **Error correction** Inform your service agency
235-0015	**Error message** Hardware error **Cause of error** Incompatible security ID of the mainboard and CCU. The mainboard and CCU have different security IDs. **Error correction** Inform your machine tool builder
235-0016	**Error message** Hardware error **Cause of error** Syscon Register check failed. The Syscon registers do not have the expected values. The hardware might be defective. **Error correction** Inform your machine tool builder
235-0017	**Error message** Hardware error **Cause of error** Initialization of the device failed. An error occurred during initialization of the device. **Error correction** Inform your machine tool builder

Error number	Description
235-0018	**Error message** Installation error **Cause of error** Could not open the file for writing. Faulty configuration. **Error correction** Inform your service agency
235-0019	**Error message** Installation error **Cause of error** Could not read the file configuration. Faulty configuration file. **Error correction** Inform your service agency
235-001A	**Error message** Installation error **Cause of error** Could not write the hardware.sys file. The configuration did not give a path for hardware.sys. **Error correction** Inform your service agency
235-001B	**Error message** Hardware error **Cause of error** An error occurred during device initialization. **Error correction** Inform your machine tool builder
235-001C	**Error message** Installation error **Cause of error** The mainboard was not detected. **Error correction** Inform your machine tool builder
235-001D	**Error message** Internal software error **Cause of error** Could not generate server interface **Error correction** Inform your service agency

Error number	Description
235-001E	**Error message** Internal software error **Cause of error** Creation of the interrupt service routine failed **Error correction** Inform your service agency
235-001F	**Error message** Internal software error **Cause of error** Interrupt does not exist **Error correction** Inform your service agency
235-0020	**Error message** Internal software error **Cause of error** Cannot process any further service functions for the given interrupt (max. 3) **Error correction** Inform your service agency
235-0021	**Error message** Internal software error **Cause of error** Undefined error **Error correction** Inform your service agency
235-0022	**Error message** Internal software error **Cause of error** Mainboard is not supported by the hardware server **Error correction** Inform your service agency
235-0023	**Error message** Hardware error **Cause of error** Unknown hardware configuration. The test for whether the processor system is single or dual failed. **Error correction** Inform your machine tool builder

Error number	Description
235-0024	**Error message** Installation error **Cause of error** Software for a single-processor system is running on a dual-processor system. Wrong software. **Error correction** Inform your machine tool builder
235-0025	**Error message** Hardware error **Cause of error** Reading the HIK failed. The hardware might be defective. **Error correction** Maschinenhersteller benachrichtigen
235-0026	**Error message** Hardware error **Cause of error** Reading the glue signature failed. The hardware might be defective. **Error correction** Inform your machine tool builder
235-0027	**Error message** Installation error **Cause of error** Unknown hardware. G50 identification failed. **Error correction** Inform your machine tool builder
235-0028	**Error message** Hardware error **Cause of error** There is no network card or there is an internal software error. Reading the MAC address failed. **Error correction** Inform your machine tool builder
235-0029	**Error message** Internal software error **Cause of error** There is a reading error from PCI base16 **Error correction** Inform your service agency

Error number	Description
235-002A	**Error message** Internal software error **Cause of error** There is a reading error from PCI base32 **Error correction** Inform your service agency
235-002B	**Error message** Installation error **Cause of error** This software does not support this control. Wrong hardware or hardware is defective. **Error correction** Inform your machine tool builder
235-002C	**Error message** Installation error **Cause of error** Wrong hardware. **Error correction** Inform your machine tool builder
235-002D	**Error message** Internal software error **Cause of error** Access to DPRAM failed **Error correction** Inform your service agency
235-002E	**Error message** Hardware will be simulated **Cause of error** The control could not find a CCU and therefore switched into the simulation mode. **Error correction** - Check the CCU - Check the connection to the CCU
235-002F	**Error message** PROFIBUS/PROFINET: Hardware error **Cause of error** The TNC cannot address the PROFIBUS/PROFINET interface. The interface is defective or incompatible with the TNC. **Error correction** Inform your service agency

Error number	Description
235-0030	**Error message** Firmware on ProfiNet interface is faulty or missing **Cause of error** The ProfiNet interface does not have any firmware. **Error correction** Inform your machine tool builder
235-0031	**Error message** Firmware inspection on ProfiNet interface failed **Cause of error** The TNC cannot open the firmware file for the ProfiNet interface. **Error correction** Inform your machine tool builder
235-0032	**Error message** Firmware update on ProfiNet interface failed **Cause of error** The TNC cannot read the firmware file for the ProfiNet interface. **Error correction** Inform your machine tool builder
235-0033	**Error message** Error in firmware file for ProfiNet interface **Cause of error** The TNC found a checksum error in the firmware file for the ProfiNet interface. **Error correction** Inform your service agency
235-0034	**Error message** Firmware update on ProfiNet interface failed **Cause of error** Error during the firmware update on the ProfiNet interface. **Error correction** Inform your machine tool builder

Error number	Description
235-0035	**Error message** ProfiNet: Hardware error **Cause of error** The TNC cannot configure the ProfiNet interface. Possible causes: - The configuration data for the ProfiNet interface are incomplete or incorrect. - The firmware of the ProfiNet interface is not compatible with the control software. **Error correction** Inform your machine tool builder
235-0036	**Error message** ProfiNet: Hardware error **Cause of error** The firmware of the ProfiNet interface is not compatible with TNC. **Error correction** Inform your machine tool builder
235-0037	**Error message** Initialization of ProfiNet interface failed **Cause of error** The TNC cannot read a ProfiNet project file. **Error correction** Inform your machine tool builder
235-0038	**Error message** Initialization of ProfiNet interface failed **Cause of error** The ProfiNet interface requests unavailable project files from the TNC. **Error correction** Inform your machine tool builder
235-0039	**Error message** CBE cannot be started **Cause of error** Unsuccessful start of the ProfiNet interface CBE30. **Error correction** Inform your machine tool builder

Error number	Description
235-003A	**Error message** Error in memory locking **Cause of error** Required memory could not be reserved. **Error correction** Inform your service agency
235-003B	**Error message** Firmware update not possible **Cause of error** Internal error during firmware update! **Error correction** Inform your machine tool builder
235-003C	**Error message** Version information missing in firmware file **Cause of error** No version information was found in a firmware file. **Error correction** Inform your machine tool builder
235-003D	**Error message** Firmware could not be started **Cause of error** Unsuccessful device firmware start. **Error correction** Inform your machine tool builder
235-003E	**Error message** Error in the hardware configuration **Cause of error** An error was found in the hardware configuration. **Error correction** Inform your machine tool builder
235-003F	**Error message** Error in ProfiNet communication **Cause of error** An error occurred during communication with a unit on the ProfiNet bus. **Error correction** - Check all devices and connections - Contact your machine tool builder

Error number	Description
235-0040	**Error message** No access to the configuration **Cause of error** Required configuration data could not be requested from configuration server. **Error correction** Inform your service agency
235-0041	**Error message** No access to event server **Cause of error** Failed access to event server. **Error correction** Inform your service agency
235-0042	**Error message** CC controller unit cannot be started **Cause of error** A CC controller unit cannot be started or the firmware could not be correctly transferred. **Error correction** - CC controller unit - Contact your machine tool builder
235-0043	**Error message** HSCI watchdog could not be deleted **Cause of error** The HSCI watchdog could not be deleted. **Error correction** Inform your service agency
235-0044	**Error message** Error in HSCI communication **Cause of error** An error occurred during communication with a unit on the HSCI bus. **Error correction** - Check all devices and connections - Inform your service agency

Error number	Description
235-0045	**Error message** HSCI connection error **Cause of error** A connection error was found on the HSCI bus. **Error correction** - Check all devices and connections - Check the connector sequence of the HSCI cable (X500 -> X502 or X501 -> X502) - Inform your service agency.
235-0046	**Error message** Error in detection of CC controller unit **Cause of error** An error occurred during detection of the connected CC controller unit. **Error correction** Inform your service agency
235-0047	**Error message** Error in message from CC **Cause of error** Erroneous information was received from a CC. **Error correction** Inform your service agency
235-0048	**Error message** Error in message from PL module **Cause of error** Erroneous information was received from a PL module. **Error correction** Inform your service agency
235-0049	**Error message** PL error **Cause of error** A PL module has reported an error. **Error correction** Inform your service agency
235-004A	**Error message** Software versions of PL module and MC do not match **Cause of error** The software versions of the PL module and the MC computer unit do not match. **Error correction** Inform your machine tool builder

Error number	Description
235-004B	**Error message** PL module could not be started **Cause of error** PL program could not be started. **Error correction** - Check the PL module - Inform your service agency
235-004C	**Error message** PL: No bus model found **Cause of error** No bus module was found in a PL module. **Error correction** - Check the PL module - Inform your service agency
235-004D	**Error message** Unknown PL software **Cause of error** A PL module has unknown software. **Error correction** - Check the PL module - Inform your service agency
235-004E	**Error message** Too many slots on PL module **Cause of error** A PL module has more slots than allowed. **Error correction** - Check the PL module - Inform your service agency
235-004F	**Error message** Error in identification of the hardware **Cause of error** A hardware unit could not be properly identified. **Error correction** Inform your service agency
235-0050	**Error message** Hardware not found **Cause of error** A required hardware unit was not found. **Error correction** - Check all devices and connections - Inform your service agency

Error number	Description
235-0051	**Error message** Error in communication with SPI module **Cause of error** An error occurred during communication with an SPI module. **Error correction** Inform your service agency
235-0052	**Error message** Error in a file operation **Cause of error** A file operation has failed. **Error correction** Inform your service agency
235-0053	**Error message** Error during hardware detection **Cause of error** Error during hardware detection **Error correction** Inform your service agency
235-0054	**Error message** Access of non-supported hardware **Cause of error** A connected hardware unit is not supported by the software in use. **Error correction** Inform your machine tool builder
235-0055	**Error message** Firmware update required (%1) **Cause of error** A firmware update is required on an assembly. The assembly concerned is indicated in the additional information. **Error correction** This firmware update requires a confirmation by the user. Note the following messages.

Error number	Description
235-0056	**Error message** Firmware update required (%1) **Cause of error** A firmware update is required on an assembly. The assembly concerned is indicated in the additional information. **Error correction** - Shut down the control software. - Manually start the firmware update on the HeROS console.
235-0057	**Error message** Firmware update is running (%1) **Cause of error** At present the control is updating the firmware on an assembly. The assembly concerned is indicated in the additional information. **Error correction** Wait until the firmware update has been completed. Note the following messages.
235-0058	**Error message** Firmware update has been completed (%1) **Cause of error** The firmware update was completed successfully. The assembly concerned is indicated in the additional information. **Error correction**
235-0059	**Error message** Firmware update failed (%1) **Cause of error** The firmware update has failed. The assembly concerned is indicated in the additional information. **Error correction** - Note further error messages. - Eliminate the cause of error. - Shut down the control and restart. The firmware update will be repeated automatically the next time the control is started.

Error number	Description
235-005A	**Error message** Firmware update failed (%1) **Cause of error** The firmware update has failed. The assembly concerned is indicated in the additional information. **Error correction** The assembly might now no longer be usable. Inform your service agency.
235-005B	**Error message** Control software shutdown delayed **Cause of error** At present the control software cannot be shut down because a firmware update is in progress. **Error correction** Wait until the firmware update has been completed. The control software then shuts down automatically.
235-0060	**Error message** Not enough main memory (RAM) **Cause of error** There is not enough working memory (RAM) on the MC to operate the control. **Error correction** Inform your service agency
235-0061	**Error message** Incompatible peripheral device (%1) **Cause of error** A peripheral device is incompatible with this control software and they cannot be operated together. The device concerned is indicated in the additional information. **Error correction** - Exchange the device - Inform your service agency

Error number	Description
235-0062	**Error message** Unknown device on HSCI bus (%1) **Cause of error** The NC software identifies every connected device by means of a device table. The table indicates whether the device is supported by the software version installed on the control. A device that the software does not support or that is not yet entered in the device table is connected to the HSCI bus. The device concerned is indicated in the additional information. The device table might have been overwritten by installing an update that was generated by an older software version. **Error correction** - Inform your service agency. - Update the NC software if the device is not supported by the currently installed version of the NC software. - Update the device table. The installed software might be able to control a new device that is not yet listed in the device table. In this case the device table must be updated.
235-0063	**Error message** Unknown device on the ProfiNet interface (%1) **Cause of error** The NC software identifies every connected devices by means of a device table. The table indicates whether the device is supported by the software version installed on the control. A device that the software does not support or that is not yet entered in the device table is connected over the ProfiNet interface. The device concerned is indicated in the additional information. **Error correction** - Inform your service agency - Run an update of the NC software if the device is not supported by the currently installed version of NC software. - Update the device table. A new device that is not yet listed in the device table might be supported by the installed software. In this case an update of the device table is required.

Error number	Description
235-0064	**Error message** Unknown device on the DriveCLiQ interface (%1) **Cause of error** The NC software identifies every connected devices by means of a device table. The table indicates whether the device is supported by the software version installed on the control. A device that the software does not support or that is not yet entered in the device table is connected over the DriveCLiQ interface. The device concerned is indicated in the additional information. **Error correction** - Inform your service agency - Run an update of the NC software if the device is not supported by the currently installed version of NC software. - Update the device table. A new device that is not yet listed in the device table might be supported by the installed software. In this case an update of the device table is required.
235-0065	**Error message** Device table is faulty **Cause of error** The device table is incorrect or cannot be used by the control. **Error correction** Inform your service agency.
235-0066	**Error message** Device (%1) reports error: %2 **Cause of error** A device connected with the control has an error. The device concerned is indicated in the additional information. Possible causes: - The device code programmed on the device is incorrect. - The device's firmware is not compatible with the control software. - The device is defective. **Error correction** Inform your machine tool builder.

Error number	Description
235-0067	**Error message** Error on device (%1) **Cause of error** An error has occurred on a device connected with the control. The device concerned is indicated in the additional information. Possible causes: - The device code programmed on the device is incorrect. - The device's firmware is not compatible with the control software. - The device is defective. **Error correction** Inform your machine tool builder.
235-0068	**Error message** Exchange the peripheral device (%1) **Cause of error** During startup it was discovered that a hardware component connected to the control is not compatible with the current level of the NC software. The device concerned is indicated in the additional information. A required update of the device's firmware is not possible. HEIDENHAIN recommends exchanging the component as soon as possible. **Error correction** Contact your service agency.
235-0069	**Error message** PROFINET: Protocol error PROFINET: Protocol error **Cause of error** A PROFINET end device has transmitted data to the control that the control cannot interpret. It could be that the version of the PROFINET protocol used by the PROFINET end device is not supported by the TNC, The PROFINET end device concerned is indicated in the additional information. **Error correction** Inform your service agency.

Error number	Description
235-006A	**Error message** Error in communication with the PROFINET controller Error in communication with the PROFINET controller **Cause of error** An error occurred in the communication between the control and the PROFINET controller. **Error correction** Inform your service agency.
235-006B	**Error message** IOC file not configured IOC file not configured **Cause of error** No file name was given for the IOC file. **Error correction** Configure the file name IOC file
235-006C	**Error message** Error in IOC file **Cause of error** There is an error in the IOC file. The additional information provides more data. **Error correction** - Check and correct the IOC file with the PC software IOconfig. - Inform your service agency.
235-006D	**Error message** Insufficient or faulty command parameters **Cause of error** A command with faulty or insufficient parameters was transferred to the part of the NC software responsible for the control of the hardware components. **Error correction** - Check the parameters - Inform your service agency.
235-006E	**Error message** IOC file cannot be opened **Cause of error** The IOC file could not be opened. The file name is indicated in the additional information. **Error correction** Check the configuration of the IOC file name.

Error number	Description
235-006F	**Error message** IOC file format error **Cause of error** Control cannot interpret the IOC file. The file format is faulty or unknown. **Error correction** Check the IOC file with the PC software IOconfig.
235-0070	**Error message** Incorrect format version of the IOC file **Cause of error** The IOC file cannot be processed because it is in the wrong format version. Minimum required format version: - PROFIBUS: IOC-V2 - AS-i: IOC-V2 - HSCI: IOC-V3 - PROFINET: IOC-V4 **Error correction** Check the IOC file and provide the correct format version.
235-0071	**Error message** IOC file: No PROFINET controller is configured **Cause of error** The PROFINET controller cannot be configured because there is no data for it in the IOC file. **Error correction** Use the PC software IOconfig to configure the PROFINET controller.
235-0072	**Error message** PROFINET: Too many devices configured **Cause of error** More PROFINET devices were configured than allowed. **Error correction** - Observe the additional information. - Reduce the number of configured devices.
235-0073	**Error message** PROFINET: project setup error **Cause of error** The configuration of the PROFINET topology is faulty. **Error correction** - Observe the additional information. - Correct the IOC file.

Error number	Description
235-0074	**Error message** PROFINET: Process data memory is insufficient **Cause of error** The maximum size of the process data for PROFINET devices has been exceeded. **Error correction** Reduce the number of PROFINET end devices or the connected modules.
235-0075	**Error message** PROFINET: No common RT class supported **Cause of error** A PROFINET end device cannot be activated by the control because the control and the end device have no common RT class. The device concerned is indicated in the additional information. **Error correction** Configure the PROFINET end device that can be operated with RT class 1.
235-0076	**Error message** PROFINET interface not activated **Cause of error** The control cannot activate the PROFINET interface because there is an error. **Error correction** - Note further messages. - Eliminate the cause of error. - Shut the control down and restart it.
235-0077	**Error message** HSCI cabling error: Controller connected to X501 **Cause of error** At least one controller unit (CC, UEC, UMC) was connected via HSCI to X501 of the MC. However, controller units must be connected to X500 of the MC. **Error correction** - Check the HSCI cabling - Connect all controller units (CC, UEC, UMC) an X500 of the MC - If the problem continues, please generate a service file and inform your service agency

Error number	Description
235-0078	**Error message** HSCI cabling error: Too many devices on X501 **Cause of error** There are too many HSCI components connected to X501 of the MC **Error correction** - Check the HSCI cabling. - Use the default values of your control's Technical Manual for the maximum number of HSCI components. - Check the HSCI configuration. It may be possible to connect HSCI components to the X500 connector of the MC. - Generate the service files and inform your service agency.
235-0079	**Error message** No contact to ProfiNet terminal **Cause of error** Interrupted communication between the control and a ProfiNet terminal. **Error correction** Check the hardware setup, the IOC file and the options set. You can find more information in the PROFINET diagnostics.
235-007A	**Error message** Faulty module configuration on ProfiNet terminal **Cause of error** The ACTUAL and NOMINAL configuration do not match on a ProfiNet terminal: - A module is configured, but it has not be plugged into the device. - A different module is configured than is plugged into the device. **Error correction** Check the hardware setup, the IOC file and the options set. You can find more information in the PROFINET diagnostics.

Error number	Description
235-007B	**Error message** Unknown device on SPI bus (%1) **Cause of error** The NC software identifies every connected devices by means of a device table. The table indicates whether the device is supported by the software version installed on the control. A device that the software does not support or that is not yet entered in the device table is connected with the SPI bus. The device concerned is indicated in the additional information. **Error correction** - Inform your service agency. - Run an update of the NC software if the device is not supported by the currently installed version of NC software. - Update the device table. A new device that is not yet listed in the device table might be supported by the installed software. In this case an update of the device table is required.
235-007C	**Error message** Error during read-in of the machine configuration data **Cause of error** The file(s) of the machine configuration or machine parameters cannot be read. It is either missing or damaged. **Error correction** - Start control as a programming station - Check the file(s) of the machine configuration (machine parameters) and, if required, recreate or correct them - Inform your service agency
235-007D	**Error message** PLCE partition has not yet been formatted **Cause of error** The encrypted PLCE partition has not yet been formatted. **Error correction** - Start the PLCE setup dialog - Enter an encryption password - Format the PLCE partition

Error number	Description
235-007E	**Error message** Encryption password for PLCE partition incorrect **Cause of error** The PLCE partition could not be integrated because the encryption password is incorrect or the partition is not yet formatted. **Error correction** - Start the PLCE setup dialog. - Enter the correct encryption password. - Include the partition. or - Enter a new encryption password. - Format the PLCE partition. - Include the partition.
235-007F	**Error message** PLCE partition cannot be integrated **Cause of error** The PLCE partition is already being used at present and therefore cannot be included. **Error correction** - Manually enable the PLCE partition or - Reboot the control
235-0080	**Error message** No password available for PLCE partition **Cause of error** An existing PLCE partition cannot be integrated for the following reasons: - Failure to read the password from the SIK. - Nor is the password available anywhere else. **Error correction** Ensure that the correct SIK is inserted in the MC.
235-0081	**Error message** Machine configuration file (.mcg) %1 is faulty **Cause of error** There is an error in the machine configuration. You will find more detailed information on the error in the additional information. **Error correction** Inform your service agency.

Error number	Description
235-0083	**Error message** Error during import of the machine configuration file (.mcg) %1 **Cause of error** The machine configuration file (.mcg) cannot be imported. You will find more detailed information on the error in the additional information. **Error correction** - Check the configuration datum CfgPlcPath.compCfgFile. - Inform your service agency.
235-0085	**Error message** Topology error in IOC file **Cause of error** The options set in the machine configuration do not match those in the IOC file. You will find more detailed information on the error in the additional information. **Error correction** - Check the options set in the machine configuration and in the IOC file. - Inform your service agency.
235-0086	**Error message** Supervisor takes over control of PROFINET terminal **Cause of error** A supervisor has taken control of a PROFINET terminal connected to the control. The machine cannot be switched back on until the supervisor has returned control. **Error correction** - Tell the supervisor to return control to the NC control - Acknowledge the error - Switch on the machine - You can find more information in the PROFINET diagnostics
235-0087	**Error message** NC software not supported by hardware **Cause of error** – The presently installed NC software version is not supported by this control hardware. - The MC main computer offers too little computing power to support all functions of the installed software. **Error correction** - Check the combination of NC software and control hardware – Inform your service agency

Error number	Description
235-0088	**Error message** Illegal parallel connection of two power modules **Cause of error** - Two different power modules were connected parallel through an adapter. - Only power modules of the same type can be connected in parallel. - The affected devices are shown in the additional information. **Error correction** - Check the connection of the power modules and correct it. - Inform your service agency.
235-008A	**Error message** Not possible to switch the HSCI data rate **Cause of error** The HSCI data rate of 1 Gbit/s was preset in the configuration datum MP_dataRateHsci in CfgHardware. However, the HSCI system cannot be operated with this data rate because - no HSCI device was connected, or - at least one connected HSCI device is not suited for a 1 Gbit/s data rate. **Error correction** Check the connected HSCI devices for suitability for a 1 Gbit/s data rate. In the configuration datum MP_dataRateHsci in CfgHardware, select the data rate 100 Mbit/s or “as fast as possible.”
235-008B	**Error message** Switchover of HSCI data rate has failed **Cause of error** The HSCI data rate of 1 Gbit/s was preset in the configuration datum MP_dataRateHsci in CfgHardware. However, the HSCI system could not be switched to the 1 Gbit/s data rate. There might be connecting cables in the HSCI system that are not made for this data rate. **Error correction** Check the HSCI cable for suitability for the 1 Gbit/s data rate. In the configuration datum MP_dataRateHsci in CfgHardware, select the data rate 100 Mbit/s or “as fast as possible.”

Error number	Description
235-008C	**Error message** Configure the HSCI data rate to 100 Mbit **Cause of error** In the configuration datum MP_dataRateHsci in CfgHardware, the HSCI data rate “as fast as possible” was selected. The control found that all connected HSCI are suited for the 1 Gbit/s data rate, but there is at least one cable in the HSCI system that is not suited for this data rate. Because of this hardware configuration, system startup is delayed. **Error correction** To speed up the system startup, do the following: In the configuration datum MP_dataRateHsci in CfgHardware, select the data rate 100 Mbit/s or check the HSCI cable for suitability for the 1 Gbit/s data rate.
235-008D	**Error message** Required HSCI data rate cannot be attained **Cause of error** There is at least one device in the HSCI system that requires a data rate of 1 Gbit/s for operation. However, this data rate cannot be set because at least one further device in the HSCI system is not suited for a data rate of 1 Gbit/s. **Error correction** Check all HSCI devices for suitability for a 1 Gbit/s data rate.
235-008E	**Error message** Required HSCI data rate cannot be attained. **Cause of error** There is at least one device in the HSCI system that requires a data rate of 1 Gbit/s for operation. However, this data rate cannot be set because at least one connecting cable in the HSCI system is not suited for a data rate of 1 Gbit/s. **Error correction** Check the HSCI connecting cable for suitability for a data rate of 1 Gbits.
235-008F	**Error message** HSCI data rate of 1 Gbit/s is required **Cause of error** The HSCI data rate of 100 Mbit/s was preset in the configuration datum MP_dataRateHsci in CfgHardware. - However, at least one connected HSCI device requires a 1 Gbit/s data rate for operation. **Error correction** In the configuration datum MP_dataRateHsci in CfgHardware, select the 1 Gbit/s setting or “as fast as possible.”

Error number	Description
235-0090	**Error message** No original HEIDENHAIN driver software **Cause of error** Software for the operation of devices of a third manufacturer has been loaded via IOconfig to the control. It is not original HEIDENHAIN software. This software is not activated. **Error correction** - Use only original HEIDENHAIN software. - Inform your service agency.
235-0091	**Error message** Driver software cannot be activated **Cause of error** Software for the operation of devices of a third manufacturer was transferred via IOconfig to the controller. Due to an error, this software cannot be activated. **Error correction** - Note the additional information - Inform your service agency
235-0092	**Error message** Not enough memory on periphery hardware (%1) **Cause of error** Since not enough memory is available on a device, that device cannot be run with this control software. The device concerned is indicated in the additional information. **Error correction** - Exchange device - Inform your service agency
235-0093	**Error message** Identification requested over PROFINET **Cause of error** A participant in a PROFINET network, usually a programming device, is demanding that the control identify itself. **Error correction**
235-0094	**Error message** Error while initializing the device %2, SN: %4 **Cause of error** The device %2 (ID number %3, serial number %4, path %1) could not be initialized. **Error correction** Inform your service agency

Error number	Description
235-0095	**Error message** Excessive propagation time in HSCI system **Cause of error** Too many HSCI participants are connected to the HSCI bus, or the total length of the HSCI cables is too long. **Error correction** - Reduce the number of HSCI participants - Use shorter HSCI cables
235-0096	**Error message** Error while evaluating a filter condition in the IOCP file **Cause of error** DEFINE missing in the MCG file. Faulty filter condition in the IOCP file. The incorrectly evaluated condition is shown in the additional data. The name of the MCG file is shown in the additional data. **Error correction** - Check the MCG file and correct it if necessary - Check the filter condition in the IOCP file and correct it if necessary
235-0097	**Error message** IOC options defined more than once **Cause of error** IOC options are defined twice in the configuration or in the MCG file. **Error correction** Correct the configuration: remove the superfluous IOC options
235-0098	**Error message** Empty IOC option defined **Cause of error** An empty IOC option is defined in the configuration or in the MCG file. **Error correction** Correct the configuration: remove the empty IOC option

Error number	Description
236-A001	**Error message** Command cannot be run **Cause of error** The ProfiNet software module cannot run a command. Possible causes: - The addressed ProfiNet end device is in a condition that prevents execution of the command. - The addressed ProfiNet end device does not exist or is not available. - The command code unknown **Error correction** Inform your service agency
236-A002	**Error message** Error during read-in of ProfiNet configuration data **Cause of error** The ProfiNet software module cannot read its configuration data. **Error correction** Check the configuration data and restart the control.
236-A003	**Error message** Initialization of ProfiNet interface failed **Cause of error** An error occurred during device initialization of the ProfiNet interface. Possible causes: - The TNC cannot speak to the ProfiNet interface. - The firmware of the ProfiNet interface is not compatible with the control software. - The configuration data for the ProfiNet interface are incorrect. **Error correction** Inform your machine tool builder.
236-A004	**Error message** Change of operating mode in ProfiNet interface failed **Cause of error** The ProfiNet interface failed to switch between the asynchronous and cyclic operating modes. Possible cause: Internal error on the ProfiNet interface **Error correction** Inform your machine tool builder

Error number	Description
236-A005	**Error message** ProfiNet: Assignment of drives incorrectly configured **Cause of error** Inconsistent assignment of the drives connected over ProfiNet with the axes managed by the TNC. The configuration data under System/ProfiNet/ParameterSets are incomplete. **Error correction** Check the configuration.
236-A006	**Error message** ProfiNet: Error during assignment of drives **Cause of error** The TNC cannot assign an axis to a drive connected over ProfiNet. Possible causes: - The configuration data under ProfiNet/ParameterSets are incorrect. - There is a fault in a drive. **Error correction** Remove the cause of error and acknowledge the error message
236-A007	**Error message** Watchdog error on the ProfiNet interface **Cause of error** The ProfiNet interface does not react to signals from the MC. Possible cause: ProfiNet interface is defective **Error correction** Inform your machine tool builder
236-A101	**Error message** No contact to ProfiNet end device %1 **Cause of error** The ProfiNet end device does not respond. Possible causes: - Error in project setup - Inconsistent configuration of the ProfiNet end device name - Interrupted connection between TNC and ProfiNet end device - There is a fault in the ProfiNet end device. **Error correction** Check the ProfiNet topology and project setup. Acknowledge the error.

Error number	Description
236-A102	**Error message** Error during access to ProfiNet end device %1 **Cause of error** The TNC cannot initialize the ProfiNet end device. **Error correction** Acknowledge the error
236-A103	**Error message** Contact lost to ProfiNet end device %1 **Cause of error** Interrupted communication between TNC and a ProfiNet end device. Possible causes: - The end device was separated from the control. - There is a fault in the end device. **Error correction** Check the ProfiNet end device. Acknowledge the error
236-A104	**Error message** Error during access to ProfiNet end device %1 **Cause of error** An error occurred during initialization of the ProfiNet end device. The TNC cannot find any configuration data for a slot on the end device, or the data are erroneous. **Error correction** Check the configuration data under System/ProfiNet/Slots. Acknowledge the error
236-A121	**Error message** Error during access to ProfiNet end device %1 **Cause of error** An error occurred during initialization of the ProfiNet end device. The TNC cannot access the configuration data of the ProfiNet end device. **Error correction** Check the configuration and acknowledge the error
236-A122	**Error message** Error during access to ProfiNet end device %1 **Cause of error** An error occurred during initialization of the ProfiNet end device. The ProfiNet end device does not support the TNC-compatible version of the ProfiDrive profile. **Error correction** Check the configuration and acknowledge the error

Error number	Description
236-A123	**Error message** Error during access to ProfiNet end device %1 **Cause of error** An error occurred during initialization of the ProfiNet end device. There are more drive objects logged in on the ProfiNet end device for cyclic data exchange with the TNC as on the ProfiNet interface. **Error correction** Check the ProfiNet project setup and acknowledge the error
236-A201	**Error message** ProfiNet: Error while initializing the drive %1 **Cause of error** The TNC cannot initialize the drive. There are no correct configuration data for this drive. **Error correction** Check Configuration data and system/ProfiNet/Slots as well as the telegram project data. Acknowledge the error
236-A202	**Error message** ProfiNet: Error while initializing the drive %1 **Cause of error** The TNC cannot initialize the drive. Possible causes: - The TNC cannot access the configuration data of the drive. - The type of drive is not supported by the TNC or is configured incorrectly on the TNC. - The TNC and drive have different telegram types configured for cyclic data exchange. - The TNC cannot access the buffer memory for fault messages. - The TNC cannot access the buffer memory for warnings. **Error correction** Check the project setup data and acknowledge the error
236-A203	**Error message** ProfiNet: Drive %1 does not react **Cause of error** The drive does not react to the control signals of the TNC. There is a fault in the drive. **Error correction** Eliminate the cause and acknowledge the error

Error number	Description
236-A204	**Error message** ProfiNet: Drive %1 does not react **Cause of error** The drive does not react. Possible causes: - The connection between the TNC and drive was interrupted. - There is a fault in the drive. **Error correction** Eliminate the cause and acknowledge the error
236-A211	**Error message** ProfiNet: Drive %1 reports a fault **Cause of error** The drive reports a fault. Watch further messages for more detailed information on the type of fault. **Error correction** Eliminate the cause and acknowledge the error
236-A212	**Error message** ProfiNet: Drive %1 reports error code %2 **Cause of error** The drive reports a fault. **Error correction** Remove the cause of error and acknowledge the message.
236-A213	**Error message** ProfiNet: Drive %1 reports warning code %2 **Cause of error** The drive is sending a warning. **Error correction** Remove the cause and acknowledge the message.
236-A221	**Error message** ProfiNet: Drive %1 cannot be switched on **Cause of error** The TNC cannot switch on the drive. Possible cause: There is a fault in the drive. **Error correction** Eliminate the cause, acknowledge the error, and switch on again.

Error number	Description
236-A222	**Error message** Error during parameter set switchover to drive %1 **Cause of error** An error occurred during a parameter set switchover on the drive. Possible cause: There is a fault in the drive. **Error correction** Eliminate the cause, acknowledge the error, and switch on again.
236-A301	**Error message** Error when homing the axis %2 **Cause of error** The TNC cannot home the axis. Possible cause: There is a fault in the drive. **Error correction** Eliminate the cause and acknowledge the error.
236-A302	**Error message** Probe failed: Axis %2 **Cause of error** An error occurred during the probing process. Possible cause: There is a fault in the drive. **Error correction** Eliminate the cause and acknowledge the error.
236-A401	**Error message** Faulty motor encoder (drive/encoder: %1) **Cause of error** The drive encoder reports an error. The error code is provided in the additional information. **Error correction** Eliminate the cause and acknowledge the error.
236-A402	**Error message** Motor encoder (drive/encoder: %1) does not react **Cause of error** The encoder does not react to the control signals of the TNC. Possible causes: - The connection between the TNC and drive was interrupted. - There is a fault in the drive controller. **Error correction** Eliminate the cause and acknowledge the error.

Error number	Description
236-A403	**Error message** Data transfer over PROFINET was interrupted **Cause of error** The PROFINET controller did not complete the process data transfer in time. **Error correction** Inform your service agency.
237-10001	**Error message** 10001 CC%2 Alarm for software test **Cause of error** - In the automatic software test an alarm was released **Error correction** - Inform your service agency
237-10003	**Error message** 10003 CC%2 System error in error memory management %4 %5 **Cause of error** - Control was not properly shut down - Hardware problem **Error correction** - Shut down the control, the switch it off and on. - Inform your service agency
237-10004	**Error message** 10004 CC%2 Active drive during switch-off process **Cause of error** - A drive was still in the feedback loop during the switch-off process - A drive was switched on during the switch-off process **Error correction** - Check the PLC program - Inform your service agency

Error number	Description
237-10005	**Error message** 10005 Drive switch-off error code: %4 **Cause of error** - Switch-off due to an external emergency stop signal - Error code: 1 = Signal -ES.A (emergency stop input on PL, MB) 2 = Signal -ES.A.HW (emergency stop input of the handwheel) 3 = Signal -ES.B (emergency stop input on PL, MB) 4 = Signal -ES.B.HW (emergency stop input of the handwheel) Further error codes: internal code **Error correction** - Check additional information from alarm message 0x10005. - Check the position of the emergency stop switch. - Inform your service agency.
237-10006	**Error message** 10006 CC%2 SPLC-RTS status change=%4, new=%5 **Cause of error** **Error correction**
237-10007	**Error message** 10007 CC%2 Follow condition for AxGrp=%4, CC=%5, MC=%6, Cause=%7 **Cause of error** - **Error correction** -

Error number	Description
237-10008	**Error message** 10008 Error in data format of ASCII command, alarm code %4 **Cause of error** - The CC controller has detected a syntax error in a transmitted ASCII command. - The faulty command was either sent by the commissioning tool TNCopt, or - the syntax of the compensation file in machine parameter compTorqueRipple is faulty. The results of adjustments through TNCopt are saved in the compensation file. **Error correction** - If TNCopt was used, repeat the affected measurement with TNCopt. - Deactivate syntax monitoring for ASCII commands with MiscCtrlFunctions bit 7 = 1. - Deactivate the compensation file in the parameter compTorqueRipple. - Inform your service agency.
237-10009	**Error message** 10009 CC%2 Follow condition for genSafe Id=%4, timer=%5, %6, %7 **Cause of error** **Error correction**
237-1000A	**Error message** 1000A CC%2 SS2 Request of IO device DeviceVariant=%4, count=%5 **Cause of error** **Error correction**
237-1000B	**Error message** 1000B CC%2 Force the S status bit - bit=%4, mode=%5 **Cause of error** **Error correction**
237-1000C	**Error message** 1000C CC%2 axis group: stop=%4 cause=%5 axis group=%6 condition=%7 **Cause of error** **Error correction**

Error number	Description
237-1000E	**Error message** 1000E CC%2 file access action=%4, %5, %6 **Cause of error** Error when accessing file Additional info 0: Action 2: Open 4: Write 5: Read 6: Delete 7: Rename 8: Directory 103: Close 250: ASYNC interface was not enabled 251: Waiting for ASYNC acknowledge from MC 252: The MC hasn't sent any hcFILE_IO (acknowledge) 253: The MC hasn't sent any ASYNC telegram 300: Not allowed during an interrupt 301: File header is corrupt Additional info 1, 2: MC error message **Error correction** - Inform your service agency
237-10010	**Error message** 10010 CC%2 ACC parameter ID=%4 Info1=%5 **Cause of error** There is an error in the ACC parameter file: - This file was created with a wrong version of TNCopt - The file contains invalid parameter data **Error correction** - Use TNCopt to generate a new ACC parameter file - Check the software version. ID=1 Option missing ID=2 Invalid axis entry ID=10 Use is possible only on hardware with limited number of axes ID=300 Missing acceleration feedforward control (MP2600) ID=301 Motor's mass moment of inertia is missing from the motor table ID=302 Constant for torque calculation is missing from the motor table - Inform your service agency

Error number	Description
237-10011	**Error message** 10011 Syntax in %4 in line %5 in column %6 **Cause of error** - Syntax error in displayed file. - Function is not supported in the displayed file with this software version **Error correction** - Check the syntax in the displayed file - Use TNCopt to regenerate the displayed file - Use appropriate machine parameters to deactivate the function - Inform your service agency
237-10013	**Error message** 10013 NOD input in %4 in line %5 is incorrect **Cause of error** - Syntax error in displayed file. - Function is not supported in the displayed file with this software version **Error correction** - The maximum number of nodes was exceeded. - Reduce the number of nodes (NOD) - Use TNCopt to regenerate the displayed file - Use appropriate machine parameters to deactivate the function - Inform your service agency
237-10014	**Error message** 10014 Invalid axis in %4 in line %5 (SAX) **Cause of error** - Syntax error in displayed file. - Axis coupling not permissible or not possible **Error correction** - Axis coupling (SAX) is not possible because the axis is on another board - Axis coupling (SAX) is not possible because the axis is deselected - Use appropriate machine parameters to deactivate the function - Use TNCopt to regenerate the displayed file - Inform your service agency

Error number	Description
237-10015	**Error message** 10015 Compensation point errors in %4 in line %5 **Cause of error** - Syntax error in the displayed file - Compensation point (NODE) not defined **Error correction** - Too many compensation points (NODx) in defined in the displayed file - Use TNCopt to regenerate the displayed file - Use appropriate machine parameters to deactivate the function - Inform your service agency
237-10016	**Error message** 10016 Max. number of blocks in %4 exceeded **Cause of error** - Syntax error in the displayed file - Maximum number of interpolation blocks has been exceeded **Error correction** - Maximum number of interpolation blocks has been exceeded - Reduce the number of interpolation blocks - Use TNCopt to regenerate the displayed file - Inform your service agency
237-10017	**Error message** 10017 Controller unit performance too low **Cause of error** - The computing performance of the controller unit is not enough for the selected function - Too many axes were configured for the controller unit **Error correction** - Deactivation of the expanded compensations - Check the power of the controller unit used - Inform your service agency
237-10018	**Error message** 10018 Compensation point errors in %4 **Cause of error** - Syntax error in the displayed file - Compensation points are incorrectly defined **Error correction** - Faulty compensation points (NODx) in the displayed file - Compensation points (NODx) have to be in ascending order - Use TNCopt to regenerate the displayed file - Inform your service agency

Error number	Description
237-10019	**Error message** 10019 Error in %4 in line %5 **Cause of error** - Syntax error in the displayed file - Faulty signal (SIGx) in the displayed line **Error correction** - The signal (SIGx) is not possible because the index is unknown - Use appropriate machine parameters to deactivate the function - Use TNCopt to regenerate the displayed file - Inform your service agency
237-1001A	**Error message** 1001A File %4 does not exist **Cause of error** Displayed file could not be opened **Error correction** - Use TNCopt to regenerate the displayed file - Edit machine parameters to deactivate the relevant function - Inform your service agency
237-1001C	**Error message** 1001C %4 1001D %4 1001E %4 18053 %4 %1 18054 %4 %1 18055 %4 %1 **Cause of error** No help text available
237-1001F	**Error message** 1001F Condensation water on supply module **Cause of error** - Temperature in electrical cabinet is too low Condensation water on supply module **Error correction** - Check the electrical cabinet temperature - Inform your service agency

Error number	Description
237-10020	**Error message** 10020 Controller unit defective: CC%2 **Cause of error** - The given CC controller unit is defective and must be exchanged. **Error correction** - Exchange the CC controller unit - Inform your service agency
237-10021	**Error message** 10021 CC%2 file access action=%4, %5, %6 **Cause of error** Warning about file access Additional data 2: Opening of file not executed Additional data 1,2: MC error message Additional data[2] = 11: EAGAIN **Error correction**
237-10025	**Error message** 10025 CC FSuC reports error %1 **Cause of error** The FSuC (Functional Safety Microcontroller) on the CC reports an error. For further information, note the following alarm messages (239-xxxx)! **Error correction**
237-10026	**Error message** 10026 CC%2: UEC dc-link current too high (I-nom: %4, I-act: %5) **Cause of error** - Excessive DC-link current of the UEC - Machine is overloaded while machining the workpiece **Error correction** - Continue working, but with less power (reduce the feed rate, replace a blunt tool, etc.) - Reduce the power being consumed simultaneously by all drives - Reduce or limit the spindle power - Reduce the spindle acceleration - Ensure that the spindle and axes accelerate at different times - Reduce the cutting depths

Error number	Description
237-10027	**Error message** 10027 CC%2: UEC dc-link voltage too high (U-min: %4, U-act: %5) **Cause of error** - DC-link voltage of the UEC is too high **Error correction** - Check the braking resistor of the UEC and exchange it if necessary - Check the wiring of the braking resistor - Check the line fuses - For operation with an optional RM regenerative module: - Check the regenerative module and exchange it if necessary - Check the wiring of the regenerative module (power grid side and DC link) - Replace the UEC if necessary
237-10028	**Error message** 10028 CC%2: UEC dc-link voltage too low (U-min: %4, U-act: %5) **Cause of error** - DC-link voltage of the UEC is too low **Error correction** - Check the wiring of the UEC - Check the 3-phase voltage supply of the UEC - Check the line fuses - Monitor for sporadic power failures
237-10029	**Error message** 10029 CC%2: UEC dc-link voltage too high with RM (U-max: %4, U-act: %5) **Cause of error** The DC-link voltage is too high despite the use of an RM regenerative module. **Error correction** - For operation with an optional RM regenerative module: - Check the regenerative module and exchange it if necessary - Check the wiring of the regenerative module (power grid side and DC link) - Check the line fuse - For operation without an RM regenerative module: - Deactivate the machine parameter uecRecoveryModule

Error number	Description
237-1002A	**Error message** 1002A CC%2: too many UM units connected to one CC **Cause of error** - Too many UM inverters are connected to the stated CC controller unit. - For each CC, the maximum number of UM inverters (or motor connections) is limited to the number of axes possible on the CC. **Error correction** - Distribute the UM inverters over other CC controller units or adapt the configuration - Remove UM inverters that aren't being used (or use one-axis modules instead of two-axis modules) - Inform your service agency
237-1002B	**Error message** 1002B CC%2: inconsistent hardware configuration **Cause of error** Correct operation of the CC controller unit is not possible. A fundamental step during hardware identification of the CC revealed an inconsistent/damaged configuration. There are two ways for such a possibility to arise: 1. Problems in the voltage supply to the CC, such as electrical contact problems, electromagnetic interference, or phenomena in the power-up sequence of the power source. 2. A hardware defect within the CC, possibly caused by problems in the voltage supply **Error correction** - Check the voltage supply, especially ribbon cables of the CC and the redundant 5V supply over X74 - Check for bent pins in the X69 box headers - Exchange the affected CC - Inform your service agency

Error number	Description
237-1002C	**Error message** 1002C CC%2: missing HFL LP transmission **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - No immediate corrective action is necessary, since no error situation has occurred yet - Recommended preventive measures: - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination
237-1002D	**Error message** 1002D CC%2 %1: notable HFL transmission; error code: %4 **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - No immediate corrective action is necessary, since no error situation has occurred yet - Recommended preventive measures: - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination

Error number	Description
237-1002E	**Error message** 1002E CC%2 %1: notable HFL transmission (ext.); error code: %4 **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - No immediate corrective action is necessary, since no error situation has occurred yet - Recommended preventive measures: - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination
237-1002F	**Error message** 1002F CC%2 %1: faulty HFL transmission; error code: %4 **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency

Error number	Description
237-10030	**Error message** 10030 CC%2 %1: faulty HFL transmission (ext.); error code: %4 **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency
237-10031	**Error message** 10031 CC%2 Schnittstellenverletzung MC-Kommando %4 Real-time-buffer transfer from integr. oscill. is beginning Current status of the axis not received The DCC calculation module was activated Tool is inconsistent 3-D simulation graphic is being recalculated... FT and FMAXT are not supported with this NC software FN15: PRINT is not supported with this NC software FN25: PRESET is not supported with this NC software
237-13000	**Error message** 13000 CC%2: inverter reports error %1 **Cause of error** UM inverter or compact inverter reports errors. For further information, note the following alarm messages (13xxx)! **Error correction**

Error number	Description
237-13003	**Error message** 13003 UM: IGBT error %1 (maximum current: %4A, phase %5) **Cause of error** - Undervoltage monitor or short-circuit monitor of an IGBT has responded in the UM inverter or UEC compact inverter. **Error correction** - Check the current controller adjustment - Check for proper motor connection and short circuit - Check the motor for an interwinding fault - Inform your service agency - Exchange the power module
237-13004	**Error message** 13004 UM: HW overcurrent shutdown %1 (maximum current: %4A, phase %5) **Cause of error** The fast hardware overcurrent monitoring in the UM inverter or UEC compact inverter has responded - Possible causes: - Current controller parameterized incorrectly - Short-circuit - Defective power module - Excessive current ripple, for example due to an unfavorable combination of motor, inverter, and PWM frequency **Error correction** - Check the current controller adjustment - Check the motor connection for a short circuit - Check the motor for an interwinding fault - If necessary, use a stronger inverter - Increase the PWM frequency - If necessary, replace defective power module - Inform your service agency

Error number	Description
237-13005	**Error message** 13005 UM: PWM nominal value faulty %1 **Cause of error** - The controller unit supplies no PWM nominal value or an invalid one for the motor, or provides the nominal value too late - Controller configuration (machine parameters) faulty - Internal software error **Error correction** - Check the controller configuration or machine parameters for PWM frequency, encoder inputs/outputs and nominal value outputs - Check the software version - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections - Inform your service agency
237-13006	**Error message** 13006 UM: Communication error in FO connection CC%2 %1 (info = %4) **Cause of error** - UM inverter or UEC compact inverter reports errors in communication via an optical fiber (HFL, HEIDENHAIN Fiber Link) with the controller unit - Info provides information about the exact cause of the error for the diagnosis **Error correction** - Check the optical fiber connections (HFL): - Is the green LED on? - Is the optical fiber plug fully engaged? - Is the tip of the optical fiber clean? - Ensure the proper bend radius - Exchange the optical fiber - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections
237-13008	**Error message** 13008 UM: Voltage monitor B CC%2 %1 (voltage ID: %4) **Cause of error** - Voltage monitoring on the inverter has responded **Error correction** - Check the voltage supply - Inform your service agency - Exchange the power module

Error number	Description
237-13009	**Error message** 13009 UM: DRIVE OFF signal is active %1 **Cause of error** - Power supply unit (UVR or UEC) reports a fault **Error correction** - Check the power supply unit - Inform your service agency
237-1300A	**Error message** 1300A UM: Heatsink temp. greater than warning thrshld. %1 (value: %4°C) **Cause of error** - Heat sink temperature has exceeded a defined warning threshold **Error correction** - Let the UM inverter or UEC compact inverter cool off - Check the fan for function and contamination
237-1300B	**Error message** 1300B UM: Heatsink temp. greater than max. value %1 (value: %4°C) **Cause of error** - Heat sink temperature has exceeded the maximum permissible value **Error correction** - Let the UM inverter or UEC compact inverter cool off - Check the fan for function and contamination
237-1300E	**Error message** 1300E UM: SW overcurrent monitoring %1 (actual value: %4Aeff) **Cause of error** - Software overcurrent monitoring in the UM inverter or UEC compact inverter has responded **Error correction** Check the current controller adjustment. - Check the motor connection for a short circuit - Check the motor for an interwinding fault - Inform your service agency - Exchange the power module

Error number	Description
237-1300F	**Error message** 1300F UM power-up test: HIK faulty CC%2 %1 %10 **Cause of error** - An error in the hardware identification key (HIK) was detected in the switch-on test of the UM converter or UEC compact inverter **Error correction** - Inform your service agency - Exchange the power module
237-1300F	**Error message** 1300F UM power-up test: internal component faulty CC%2 %1 %10 **Cause of error** - An inadmissible internal component identifier (FPGA ID) was detected or the component could not be accessed during the switch-on test of the UM or UEC compact inverter **Error correction** - Inform your service agency - Check the software version
237-1300F	**Error message** 1300F UM power-up test: analog interface faulty CC%2 %1 %10 **Cause of error** - During the switch-on test of the UM or UEC compact inverter, it was determined that an analog interface in the inverter is not functioning properly **Error correction** - Restart the control - Inform your service agency
237-1300F	**Error message** 1300F UM power-up test: temp. measurement not possible CC%2 %1 %10 **Cause of error** - It was not possible to read the values of the heat sink temperature sensor in the switch-on test of the UM or UEC compact inverter - Sensor or associated interface (I2C) defective **Error correction** - Inform your service agency

Error number	Description
237-1300F	**Error message** 1300F UM power-up test: memory component (FRAM) defective CC%2 %1 %10 **Cause of error** - The FRAM could not be identified in the switch-on test of the UM converter or UEC compact inverter **Error correction** - Restart the control - Inform your service agency
237-1300F	**Error message** 1300F UM power-up test: serial interface (SPI) faulty CC%2 %1 %10 **Cause of error** - During the switch-on test of the UM or UEC compact inverter, it was determined that the serial interface (SPI) to the FSuC in the inverter is not functioning properly **Error correction** - Restart the control - Inform your service agency
237-1300F	**Error message** 1300F UM power-up test: meas. of current faulty CC%2 %1 %10 (info = %5) **Cause of error** - During the switch-on test of the UM or UEC compact inverter, it was determined that the measurement of current in the inverter is faulty - When the drive has been switched off, the permissible maximum offset current has been exceeded in one or more phases: (info is displayed decimally but is to be interpreted as binary) - Bit 0: Phase U - Bit 1: Phase V - Bit 2: Phase W **Error correction** - Inform your service agency

Error number	Description
237-1300F	**Error message** 1300F UM power-up test: temp. value faulty CC%2 %1 %10 (value=%5°C) **Cause of error** - In the switch-on test of the UM converter or UEC compact inverter, a heat sink temperature outside the permitted range was read out - Heat sink too hot - Sensor or associated interface defective **Error correction** - Let the device cool off - Inform your service agency
237-1300F	**Error message** 1300F UM power-up test: fan defective CC%2 %1 %10 **Cause of error** During the switch-on test of the UM inverter or UEC compact inverter, it was determined that the fan for cooling of the electronics in the inverter is not functioning properly. **Error correction** - Check the fan - Exchange the inverter - Inform your service agency
237-1300F	**Error message** 1300F UM power-up test: internal lead defective CC%2 %1 %10 **Cause of error** During the switch-on test of the UM inverter or the UEC compact inverter, it was determined that an internal lead to the FSuC (Functional Safety Microcontroller) is defective. **Error correction** - Exchange the inverter - Inform your service agency
237-13012	**Error message** 13012 UM: WD monitoring has responded %1 (received:%4, expected:%5) **Cause of error** - UM inverter or UEC compact inverter reports that watchdog on controller unit is no longer being updated **Error correction** - Restart the control - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections - Inform your service agency

Error number	Description
237-13014	**Error message** 13014 UM: Optical waveguide connection faulty %1 **Cause of error** - UM inverter or UEC compact inverter reports faults in the FO connection (fiber optic connection between controller unit and inverter) **Error correction** - Check the optical fiber connection: - Is the green LED on? - Is the optical fiber plug fully engaged? - Is the tip of the optical fiber clean? - Ensure the proper bend radius - Exchange the optical fiber
237-13015	**Error message** 13015 UM: Erroneous heatsink temperature value %1 **Cause of error** - The UM inverter or UEC compact inverter reports errors when accessing the I2C bus for reading the heat sink temperature sensor - No temperature sensor is connected or the connection is faulty - Temperature sensor is defective - I2C controller is defective **Error correction** - Restart the control - Inform your service agency
237-13016	**Error message** 13016 UM: Invalid UM debug channel %4 %1 **Cause of error** - Invalid UM debug signal selected in the oscilloscope **Error correction** - Select another signal
237-13017	**Error message** 13017 UM: Erroneous heatsink temperature value %1 **Cause of error** - The UM inverter or UEC compact inverter reports errors when accessing the I2C bus for reading the heat sink temperature sensor - No temperature sensor is connected or the connection is faulty - Temperature sensor is defective - Improper function of I2C controller **Error correction** - Restart the control - Inform your service agency

Error number	Description
237-1301A	**Error message** 1301A UM: FO signal weaker than warng threshld CC%2 %1 (value=-%4dB) 10023 CC%2: X%6 FO sign. strgth below warng threshld %1 (value = -%4dB) **Cause of error** - The signal strength of the FO connection (fiber optic cable to the UM converter) has fallen below a defined warning threshold **Error correction** - Check the optical fiber connections: - Is the green LED on? - Is the optical fiber plug fully engaged? - Is the tip of the optical fiber clean? - Ensure the proper bend radius - Exchange the optical fiber
237-1301B	**Error message** 1301B UM: FO signal weaker than minimum value CC%2 %1 (value=-%4dB) 10024 CC%2: X%6 FO signal weaker than minimum value %1 (Wert = -%4dB) **Cause of error** - The signal strength of the fiber optic connection (fiber optic cable to the UM converter) has fallen below a permissible minimum value **Error correction** - Check the fiber optic connection: - Is the green LED on? - Is the optical fiber plug fully engaged? - Is the tip of the optical fiber clean? - Ensure the proper bend radius - Exchange the optical fiber

Error number	Description
237-1301C	**Error message** 1301C UM: communication fault CC%2 %1, error code=%4 **Cause of error** The HFL communication component of the inverter reports an error Possible causes: - Electromagnetic disturbances - Hardware defective - Optical fibers (HFL) not connected correctly - Insufficient bend radius of the optical fibers (HFL) **Error correction** - Create a service file (error code gives information about the exact cause of the error for the diagnosis at HEIDENHAIN) - Inform your service agency - If required, exchange the hardware - Use TNCdiag to check the damping of the fiber-optic connections (HFL). If the damping is too high: - Check the optical fiber connections - Comply with the guidelines in the Technical Manual regarding the cable routing - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections
237-1301D	**Error message** 1301D UM: logical disturbance of data reception CC%2 %1, error code=%4 **Cause of error** The communication component for the optical fiber connection (HFL) to the inverters reports an error Possible causes: - Hardware defective - Electromagnetic disturbances - Optical fibers (HFL) not connected correctly - Insufficient bend radius of the optical fibers (HFL) **Error correction** - Create a service file (error code gives information about the exact cause of the error for the diagnosis at HEIDENHAIN) - Inform your service agency - If required, exchange the hardware - Use TNCdiag to check the damping of the fiber-optic connections (HFL). If the damping is too high: - Check the optical fiber connections - Comply with the guidelines in the Technical Manual regarding the cable routing - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections

Error number	Description
237-1301E	**Error message** 1301E UM: physical disturbance of data reception CC%2 %1, error code=%4 **Cause of error** The communication component for the optical fiber connection (HFL) to the inverters reports an error Possible causes: - Hardware defective - Electromagnetic disturbances - Optical fibers (HFL) not connected correctly - Insufficient bend radius of the optical fibers **Error correction** - Create a service file (error code gives information about the exact cause of the error for the diagnosis at HEIDENHAIN) - Inform your service agency - If required, exchange the hardware - Use TNCdiag to check the damping of the fiber-optic connections (HFL). If the damping is too high: - Check the optical fiber connections - Comply with the guidelines in the Technical Manual regarding the cable routing - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections
237-1301F	**Error message** 1301F UM: Inverter not ready %1 (info = %4) **Cause of error** - Since a UM converter or UEC compact inverter is not ready, it is not possible to switch on the drive - Reason for lack of inverter readiness: (information is shown decimally. Please interpret as binary) - Bit 0: "STO.A.P.x" - Bit 1: "STO.B.H.P.x" - Bit 2: "STO.B.L.P.x" - Bit 3: Signal for PWM release not set - Bit 4: Error in switch-on test - Bit 5: Drive motor is not (fully) configured **Error correction** - Inform your service agency - Exchange the power module if necessary

Error number	Description
237-13020	**Error message** 13020 UM: Readiness was lost %1 (info = %4) **Cause of error** - UM inverter or UEC compact inverter was switched off during operation - Reason for lack of inverter readiness: (information is shown decimally. Please interpret as binary) - Bit 0: "STO.A.P.x" - Bit 1: "STO.B.H.P.x" - Bit 2: "STO.B.L.P.x" - Bit 3: Signal for PWM release not set - Bit 4: Error in switch-on test **Error correction** - Check the entry in MP_delayTimeSTOatSS1 and increase the value if necessary (the parameterized time here must be greater than the value in MP_vCtrlSwitchOffDelay) - Inform your service agency - Exchange the power module if necessary
237-13021	**Error message** 13021 UM: Faulty Include file CC%2 %1 **Cause of error** - Software from CC controller unit, UM inverter or UEC compact inverter have not been compiled with the same include file. **Error correction** - Check the software version and update it if necessary - Inform your service agency
237-13025	**Error message** 13025 UM: Stack overflow CC%2 %1 **Cause of error** - Internal software error on the UM inverter or UEC compact inverter **Error correction** - Inform your service agency - Check the software version
237-13026	**Error message** 13026 UM: IRQ stack overflow CC%2 %1 **Cause of error** - Internal software error on the UM inverter or UEC compact inverter **Error correction** - Inform your service agency - Check the software version

Error number	Description
237-13029	**Error message** 13029 UM: Temperature sensor provides invalid measured values %1 1302D UM: Temperature sensor provides invalid measured values %1 **Cause of error** The temperature sensor on the heat sink of the UM inverter or UEC compact inverter provides invalid measured values: - No temperature sensor is connected or the connection is missing - Temperature sensor is defective **Error correction** - Restart the control - Inform your service agency
237-1302A	**Error message** 1302A UM: leakage current too high %1 **Cause of error** Isolation problem (e.g. defective motor, contamination within the inverter, humidity) **Error correction** - Replace the motor of the affected axes or check for a ground fault - Replace the power cable of the affected axes or check for a ground fault - Replace the inverter of the affected axes or check for a ground fault - Inform your service agency
237-1302C	**Error message** 1302C UM: Test software loaded **Cause of error** The inverter has a software version that has not been released yet, and has no valid checksum - This software has neither been tested nor released **Error correction** - After acknowledging the error message you can use this software for test purposes - Check the software version - Create service files - Contact your service agency

Error number	Description
237-1302E	**Error message** 1302E UM: Controller software timeout %1 1302F UM: Controller software timeout %1 **Cause of error** - The time monitor of the inverter software is reporting an exceedance - Internal software error **Error correction** - Inform your service agency
237-13032	**Error message** 13032 UM3xx must be exchanged or rebuilt %1 (Ser-Nr.: %4) **Cause of error** Umrichter mit Bremsschaltung ohne internen Pullup-Widerstand werden ab 27.05.2019 nicht mehr unterstützt **Error correction** Hardware umbauen oder tauschen (Bei Fragen an Georg Zehentner (Tel. 1845) wenden)
237-13033	**Error message** 13033 UM FSuC reports error CC%2 %1 %10 **Cause of error** The FSuC (Functional Safety Microcontroller) on the inverter reports an error. For further information, note the following alarm messages (239-xxxx)! **Error correction**
237-13034	**Error message** 13034 UM: Parameters for the thermal model are missing %1 (Ser-Nr.: %4) **Cause of error** Umrichter ohne HIK-Parameter für das thermische Modell werden bald nicht mehr unterstützt **Error correction** HIK umprogrammieren lassen
237-13035	**Error message** 13035 UM: barrier layer temp. > warning threshold %1 (value: %4°C) **Cause of error** The calculated barrier layer temperature has exceeded a defined warning threshold. **Error correction** - Reduce the profile load - Let the UM inverter or UEC compact inverter cool off - Check the fan for function and contamination

Error number	Description
237-13036	**Error message** 13036 UM: barrier layer temp. > maximum value %1 (value: %4°C) **Cause of error** The calculated barrier layer temperature has exceeded the maximum permissible value. **Error correction** - Reduce the profile load - Let the UM inverter or UEC compact inverter cool off - Check the fan for function and contamination
237-1303C	**Error message** 1303C UM: fan speed too low CC%2 %1 **Cause of error** The speed of the fan for cooling of the electronics in the inverter has fallen below the monitoring threshold. **Error correction** - Check the fan - Exchange the inverter - Inform your service agency
237-1303D	**Error message** 1303D UM: short circuit of brakes was detected %1 (Info: %4) **Cause of error** - The inverter detected a short circuit in the motor brake. - Info provides information about the exact cause of the error: Info = 1: Short circuit between BR+ and BR- Info = 2: Short circuit between BR+ and housing **Error correction** Warning: Hanging axes cannot be supported under certain circumstances. The axis can drop. Do not enter the area of danger under the axis! - Move the axis to a safe position before power-off - Inform your service agency - Check controls for motor brakes - Exchange motor
237-1303F	**Error message** 1303F UM: faulty FSuC Include file CC%2 %1 **Cause of error** The interface versions of internal components (SOC and FSuC) on the inverter do not match. **Error correction** - Check the software version and run an update if necessary - Inform your service agency

Error number	Description
237-13040	**Error message** 13040 UM: too many faulty CC telegrams in sequence %1 **Cause of error** Too many telegrams in sequence were lost during transmission of the nominal voltage values from the CC to the UM. Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency
237-13041	**Error message** 13041 UM: too many faulty CC telegrams during the monitoring period %1 **Cause of error** Too many telegrams were lost during transmission of the nominal voltage values from the CC to the UM during the monitoring period. Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency

Error number	Description
237-13042	**Error message** 13042 UM: excessive error frequency of CC telegrams %1 **Cause of error** The frequency of faulty telegrams during transmission of the nominal voltage values from the CC to the UM is above the limit value. Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency
237-13043	**Error message** 13043 UM: conspicuous CC telegrams during the monitoring period %1 **Cause of error** There are peculiar telegrams during transmission of the nominal voltage values from the CC to the UM. Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency

Error number	Description
237-13044	**Error message** 13044 UM: conspicuous CC telegrams outside the monitoring period %1 **Cause of error** There are peculiar telegrams during transmission of the nominal voltage values from the CC to the UM. Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency
237-13045	**Error message** 13045 UM: overload / IGBT (emergency stop) %1 (%4°C) 13046 UM: overload / IGBT (STO) %1 (%4°C) 13035 UM: overload / IGBT (warning) / %1 (%4°C) 13036 UM: overload / IGBT %1 (%4°C) 1300A UM: overload / heat sink (warning) %1 (%4°C) 1300B UM: overload / heat sink %1 (%4°C) **Cause of error** An excessive temperature of the power electronics or the heat sink was detected in the UM inverter or the UEC compact inverter. Possible causes: - Excessive cutting power - Excessive feed rate - Excessive continuous load - Axis moved against an obstacle or limit stop - Holding brake of axis applied during machining - Excessive temperature in the electrical cabinet (cooling failed) - Excessive axis or spindle acceleration - Fan of the UM inverter or UEC compact inverter defective

Error number	Description
237-13048	**Error message** 13048 UM3: sequencer sync pulse outside the tolerance range %1 **Cause of error** The synchronization pulse for an internal sequencer in the inverter was (too often) outside of the specified tolerance range. Electromagnetic noise or faulty electrical contacts: - of the 24 V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency
237-13049	**Error message** 13049 UM: missing HFL LP transmission **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - No immediate corrective action is necessary, since no error situation has occurred yet - Recommended preventive measures: - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination

Error number	Description
237-1304A	**Error message** 1304A UM: notable HFL transmission CC%2 %1; error code: %4 **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - No immediate corrective action is necessary, since no error situation has occurred yet - Recommended preventive measures: - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination
237-1304B	**Error message** 1304B UM: notable HFL transmission (ext.) CC%2 %1; error code: %4 **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - No immediate corrective action is necessary, since no error situation has occurred yet - Recommended preventive measures: - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination

Error number	Description
237-1304C	**Error message** 1304C UM: faulty HFL transmission CC%2 %1; error code: %4 **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency
237-1304D	**Error message** 1304D UM: faulty HFL transmission (ext.) CC%2 %1; error code: %4 **Cause of error** Possible causes: Electromagnetic noise or faulty electrical contacts: - of the 24V supply voltage - of the DC-link voltage - in the motor cabling - in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing and clamping, and also for contamination - Exchange the UM3 - Exchange the CC3 - Inform your service agency
237-137FF	**Error message** 137FF UM: Alarm %1 module = %4 line = %5 **Cause of error** - Internal error in the UM inverter or UEC compact inverter **Error correction** - Inform your service agency

Error number	Description
237-13800	**Error message** 13800 UM (FS.B):CRC error in FS communication %1 (nom.:%4,act.:%5) **Cause of error** - Checksum (CRC) in cyclic communication with SKERN-CC is faulty. **Error correction** - Inform your service agency
237-13801	**Error message** 13801 UM (FS.B):FS communication telegram counter %1 nom.:%4 act.:%5 **Cause of error** - Telegram counter in cyclic communication with SKERN-CC is faulty. **Error correction** - Inform your service agency
237-13802	**Error message** 13802 UM (FS.B):UM-DriveID error in FS communicatn. %1 nom.:%4 act.:%5 **Cause of error** - UM.driveID contains errors in cyclic communication with SKERN-CC **Error correction** - Inform your service agency
237-13803	**Error message** 13803 UM (FS.B):Error during deactivation of the FS communication %1 **Cause of error** - Error during deactivation of the FS communication with the UM or UEC - The signals STO.B and SBC.B were not set during the deactivation of the axis. **Error correction** - Drives must be turned off before deactivating an axis. - Check the (S)PLC program, and adapt it if necessary.

Error number	Description
237-13804	**Error message** 13804 UM (FS.B):Topology checking of UM parameters failed %1 **Cause of error** - Received parameters do not match the topology of the setup - Configuration of the machine parameters MP_hsciCcIndex, MP_inverterInterface, or MP_motorConnector is incorrect **Error correction** - Check the configuration of MP_hsciCcIndex, MP_inverterInterface, and MP_motorConnector, and adapt if necessary - Inform your service agency
237-13805	**Error message** 13805 UM (FS.B): FS configuration data were falsified %1 **Cause of error** - UM parameters were corrupted and do not match the calculated checksum (UM.DRIVE-ID) **Error correction** - Reboot of the control - Inform your service agency
237-13806	**Error message** 13806 UM (FS.B): UM-DriveID from UM(FS.A) and UM(FS.B) are unequal %1 **Cause of error** - Calculated checksums (UM-DRIVE.ID) for UM parameters between UM(FS.A) and UM(FS.B) are different **Error correction** - Inform your service agency
237-13807	**Error message** 13807 UM (FS.B):Error during the parameterization of the UM (FS.A) %1 **Cause of error** - Faulty data transfer between UM(FS.A) and UM(FS.B). - The received data were corrupted on the bus line or recognized as invalid by the UM(FS.A). **Error correction** - Inform your service agency - If necessary, replace the inverter

Error number	Description
237-13808	**Error message** 13808 UM (FS.B): reconfiguration of UM3 parameters not possible %1 **Cause of error** - Safety-relevant parameters of the inverter were changed (delayTimeSTOatSS0/SS1, delayTimeSBCatSS0/SS1 or driveOffGroup) **Error correction** - Restart the control in order to apply the new parameters
237-13809	**Error message** 13809 UM (FS.B):Switch-on readiness lacking %1 1:%4 2:%5 3:%6 4:%7 5:%8 **Cause of error** - The conditions for the switch-on readiness of the UM are not given: -- Info1 = 1: No valid configuration received for this axis -- Info2 = 1: SS0 stop reaction active for this axis -- Info3 = 1: Self-test for this axis has not yet run without error -- Info4 = 1: STEST.PERMIT set as release for the UM self-test -- Info5 = 1: Restart prevented due to the previous SS0 and SS1F **Error correction**
237-1380A	**Error message** 1380A UM (FS.B): CRC error in PAE communication %1 (nom.:%4,act.:%5) **Cause of error** - Checksum (CRC) in cyclic communication with PAE is faulty. **Error correction** - Inform your service agency
237-1380B	**Error message** 1380B UM (FS.B): PAE communication telegram counter %1 nom.:%4 act.:%5 **Cause of error** - Telegram counter in cyclic communication with PAE is faulty. **Error correction** - Inform your service agency

Error number	Description
237-1380C	**Error message** 1380C UM (FS.B): error during deactivation, %1 not in safe state **Cause of error** - The axis is to be deactivated, but the axis is not in a safe state - STO and SBC are still enabled **Error correction** - Put the axis in a safe state before deactivating it
237-13820	**Error message** 13820 UM (FS.B): 3.3 V (FS.A) voltage monitor exceeded CC %2 %1 **Cause of error** - The internal voltage monitor found that the 3.3 V voltage of the FS.A is too high **Error correction** Restart the control; if the error occurs again, then: - Check the X76 jumper plugs - Replace device - Inform your service agency
237-13821	**Error message** 13821 UM (FS.B): 3.3V(FS.A) voltage monitor below limit CC %2 %1 **Cause of error** - The internal voltage monitor detected that the 3.3 V voltage of the FS.A is too low **Error correction** Restart the control; if the error occurs again, then: - Check the X76 jumper plugs - Replace device - Inform your service agency
237-13822	**Error message** 13822 UM (FS.B): 5V voltage monitor exceeded CC%2 %1 **Cause of error** - The internal voltage monitor detected that the 5 V voltage is too high **Error correction** Restart the control; if the error occurs again, then: - Check the X76 jumper plugs - Replace device - Inform you service agency

Error number	Description
237-13823	**Error message** 13823 UM (FS.B): 5V voltage monitor below limit CC%2 %1 **Cause of error** - The internal voltage monitor found that the 5 V voltage is too low **Error correction** Restart the control; if the error occurs again, then: - Check the X76 jumper plugs - Replace device - Inform your service agency
237-13824	**Error message** 13824 UM (FS.B): Watchdog FS.A has responded CC%2 %1 **Cause of error** - UM inverter or UEC compact inverter reports that watchdog of the A-channel (FSuC) is no longer being updated **Error correction** - Restart the control - Inform your service agency
237-13825	**Error message** 13825 UM (FS.B): Temp. greater than warning threshold CC %2 %1 (%4°C) **Cause of error** - Processor temperature has exceeded a defined warning threshold **Error correction** - Let the UM inverter or UEC compact inverter cool off - Check the fan for function and contamination
237-13826	**Error message** 13826 UM (FS.B): Temp. greater than maximum value CC%2 %1 (%4°C) **Cause of error** - Processor temperature has exceeded the maximum permissible value **Error correction** - Let the UM inverter or UEC compact inverter cool off - Check the fan for function and contamination

Error number	Description
237-13827	**Error message** 13827 UM (FS.B): Error in the safety self-test CC%2 %1 %10 Info0: %4 **Cause of error** - Error during FS self-test. Info 0: 0 = Emergency stop during the self-test 1 = Test-initialization 2 = Test of the brake control, part 1 3 = Test of the brake control, part 2 4 = Test of the brake control, part 3 5 = Test of the STO cut-out signal, part 1 6 = Test of the STO cutout signal, part 2 7 = Test of the STO cutout signal, part 3 8 = Test of the PWM pulse inhibitor, positive test 9 = Test of the PWM pulse inhibitor, low-side negative test 10 = Test of the PWM pulse inhibitor, high-side negative test 11 = Test of the watchdog, channel A 12 = Test of the watchdog, channel B 13 = Test of the voltage monitor, channel B lower limit 14 = Test of the voltage monitor, channel B upper limit 15 = Test of the voltage monitor, channel A 16 = Test of temperature value measurement, channel B 17 = Test of the SS0 stop request 18 = Test of the SS1 stop request 19 = Test of the timers, channel A timer 1 20 = Test of the timers, channel A timer 2 21 = Test of the timers, channel B 22 = Not all required tests have been performed **Error correction** - When Info 0 = 0: If necessary, unlock emergency stop, and restart the control. - When Info 0 = 2-4: Error of the brake control. Inspect the wiring of the holding brakes. - When Info 0 = 5-21: Inform your service agency. If required, exchange the hardware. - When Info 0 = 1/22: Inform your service agency.
237-13828	**Error message** 13828 UM (FS.B): SS1F request from UM(FS.A) CC%2 %1 **Cause of error** -SS1F request (REQ.SS1F signal) from UM(FS.A) active. **Error correction** Note further pending error messages.

Error number	Description
237-13831	**Error message** 13831 UM (log): PAE test returns rising edge STEST_OK 13830 UM (log): SS1 request from PAE 6800 CC (Dbg): test code 6810 CC (Dbg): switch-off position invalid, outside the limit switches 6820 CC (log): autotest info 6840 CC (log): timer info for monitoring the calling of the brake test 6850 CC (log): LSV2 test command for FS error injection 13027 UM (log): stack overflow early warning CC%2 %1 13028 UM (log): IRQ stack overflow early warning CC%2 %1
237-13832	**Error message** 13832 UM (FS.B): Faulty start condition for PAE test %1 Info0:%4 **Cause of error** The condition for starting the self-test of the PAE module is not given on the B channel side: At the time of the self-test, the drive must be set to STO and SBC. Info 0 describes the cause bit-encoded - Bit 0: STO.B.H not active - Bit 1: STO.B.L not active - Bit 2: SBC.B not active **Error correction** - Stop the drive before the start of the self-test (STO and SBC must be active) - Inform your service agency

Error number	Description
237-13833	**Error message** 13833 UM (FS.B): Faulty start condition for self-test %1 Info0:%4 **Cause of error** The condition for starting the self-test of the UM is not given At the time of the self-test, the drive must be set to STO and SBC. Further conditions must also be met. Info 0 describes the cause bit-encoded - Bit 0: STEST.Permit missing - Bit 1: STO.A not active - Bit 2: STO.B.H not active - Bit 3: STO.B.L not active - Bit 4: SBC.A not active - Bit 5: SBC.B not active - Bit 6: SEU error occurred - Bit 7: STEST.OK of PAE missing **Error correction** - Stop the drive before the start of the self-test (STO and SBC must be active) - Check the (S)PLC program, and adapt it if necessary - Inform your service agency
237-13834	**Error message** 13834 UM (FS.B): PAE reports internal error (-REQ.SS1F) %1 **Cause of error** The PAE-H PL module reports an internal error (switch-off due to -REQ.SS1F) Possible causes: - Maximum temperature exceeded - Supply voltage not correct - PAE module not connected correctly - Internal PAE error **Error correction** - Check the expanded information in the bus diagnostics - Stay within the temperature range - Check the power supply - Exchange the PAE-H module - Inform your service agency
237-13835	**Error message** 13835 UM (FS.B):Temperature below warning threshold CC %2 %1 (%4°C) **Cause of error** The processor temperature has fallen below a defined warning threshold. The ambient temperature in the electrical cabinet must be greater than +1°C. **Error correction** - Check the temperature conditions in the electrical cabinet - Exchange the hardware - Inform your service agency

Error number	Description
237-13836	**Error message** 13836 UM (FS.B):Temperature lower than min. value CC%2 %1 (%4°C) **Cause of error** The processor temperature has fallen below the permissible minimum value. The ambient temperature in the electrical cabinet must be greater than 0°C. **Error correction** - Check the temperature conditions in the electrical cabinet - If present: Check the climate control unit - Exchange the hardware - Inform you service agency
237-13837	**Error message** 13837 UM (FS.B): brake control faulty CC%2 %1 %10 Info0: %4 **Cause of error** The test of the brake control detected a fault. Refer to Info0 for the possible cause. Info 0: 1 = Brake configured but not detected 2 = Short-circuit of the high-side or of the brake with 24V 3 = Short-circuit of the low-side or of the brake with 0V 4 = Short-circuit of the high-side with 24V and short-circuit of the low-side with 0V 5 = Supply voltage too low (< 23.75 V) **Error correction** - Check the supply voltage (23.75 V to 26.25 V) - Check the wiring of the brake - Inform your service agency
237-13838	**Error message** 13838 UM (FS.B): An SS0 or SS1F is preventing a restart %1 **Cause of error** - A previous SS0 or SS1F reaction of the inverter is preventing the drive from being switched on. **Error correction** - Restart the control.

Error number	Description
237-13839	**Error message** 13839 UM (FS.B): unreleased testing software loaded: CC%2 %1 **Cause of error** An unreleased test software is installed on the inverter: - This software has neither been tested nor officially released - No checksum will be calculated for this software This software is intended solely for testing purposes! **Error correction** This software or firmware must be replaced by a software or firmware that has been officially released: - Create service files - Inform your service agency
237-1383A	**Error message** 1383A UM: Checksum error in the program code CC%2 %1 **Cause of error** - A checksum error was detected in the program code of the inverter. - Inverter defective. **Error correction** - Inform your service agency - If required, exchange the hardware
237-13890	**Error message** 13890 UM (FS.B): faulty FS Include file CC%2 %1 (act: %4 nom: %5) **Cause of error** - SKERN-CC and UM (FS.B) software have not been compiled with the same Include file. **Error correction** - Check the NC software version, and reinstall it if necessary - Inform your service agency
237-13F00	**Error message** 13F00 UMFSSW: error while deactivating axis %1 **Cause of error** Deactivation of a safe axis is not supported by the functional safety (SKERN-CC). **Error correction** Restart the control without deactivating the safe axis

Error number	Description
237-17FFC	**Error message** 17FFC Axis %1: PLC module 9311 no longer supported. **Cause of error** This function is no longer supported by this software version. **Error correction** Inform your service agency
237-17FFD	**Error message** 17FFD CC %2 controller clock slave axis %1 not identical to master axis **Cause of error** The parameters MP_ctrlPerformance and MP_ampPwmFreq are parameterized differently for the master and slave. **Error correction** - Check the entries in MP_ctrlPerformance and MP_ampPwmFreq, and set them to the same values for master and slave. - Inform your service agency.
237-17FFE	**Error message** 17FFE Axis %1: Measurement canceled **Cause of error** - Note further messages. **Error correction** - Repeat the measurement.
237-17FFF	**Error message** 17FFF Axis %1: Amplitude of excitation signal is too high **Cause of error** Limit of provided current or voltage reached during the measurement. **Error correction** - Reduce the amplitude of the excitation signal
237-18000	**Error message** 18000 CC %2: Not enough memory available. **Cause of error** - The memory requirements for the requested measurement are too high. **Error correction** - Reduce the recording time. - Reduce the recording rate.

Error number	Description
237-18001	**Error message** 18001 Axis %1 Alarm for software test **Cause of error** - In the automatic software test an alarm was released **Error correction** - Inform your service agency
237-18003	**Error message** 18003 kv factor of the cmp file is not equal to MP %1 **Cause of error** - The kv factors in a cmp file and parameter file are different **Error correction** - Change the kv factor (kvfactor) in the machine parameter file to the value from the cmp file (compTorqueRipple) - Inform your service agency.
237-18004	**Error message** 18004 ki factor of the cmp file is not equal to MP %1 **Cause of error** - The ki factors in a cmp file and machine parameter file are different **Error correction** - Change the ki factor (vCtrlIntGain) in the parameter file to the value from the cmp file (compTorqueRipple) - Inform your service agency.
237-18006	**Error message** 18006 DQ-ALM %1: Uz nominal value too low **Cause of error** - The DC-link voltage of a DRIVE-CLiQ ALM power supply as defined in the machine parameter "ampBusVoltage" is smaller than the rectified line voltage. **Error correction** - Check the entry in machine parameter "ampBusVoltage." - Inform your service agency.

Error number	Description
237-18007	**Error message** 18007 EnDat para. invalid: axis %1code:%4 value: %5 **Cause of error** - During the initialization of the EnDat motor encoder for the given axis, a parameter value was detected that is not supported. - The code of the error message describes the cause: 100: EnDat 2.1 Parameters of the encoder manufacturer, word 20/21: "Measuring step or measuring steps per revolution" = 0 is not supported by the controller software if it is needed to calculate the commutation of a motor. 101: EnDat 2.1 Parameters of the encoder manufacturer, word 17: "Number of distinguishable revolutions" is supported only up to 65534. 102: EnDat 2.1 Parameters of the encoder manufacturer, word 13: CC424 "Number of clock pulses to transmit the position value (transmission format)" is supported up to 32. 200: EnDat 2.2 Parameters of the encoder manufacturer, word 4: "Scaling factor for temperature" unknown. **Error correction** - Exchange the encoder - Inform your service agency
237-18008	**Error message** 18008 EnDat parameter invalid in axis:%1 Code:%4 Value:%5 **Cause of error** - During the initialization of the EnDat position encoder for the given axis, a parameter value was detected that is not supported. - The code of the error message describes the cause: 101: EnDat 2.1 Parameters of the encoder manufacturer, word 17: "Number of distinguishable revolutions" is supported only up to 65534. 200: EnDat 2.2 Parameters of the encoder manufacturer, word 4: "Scaling factor for temperature" unknown. **Error correction** - Exchange the encoder - Inform your service agency.

Error number	Description
237-18009	**Error message** 18009 Communic. error, EnDat motor enc. %1, err. code %4 **Cause of error** - An error occurred with the EnDat motor encoder - The error code describes the cause: - 101 and 102: The encoder generated an internal error message. A further alarm 0x1800F describes the exact cause. - 103, 104 and 105: An error occurred in communication. Possible causes: - Defective encoder line - Encoder line is not suitable for digital EnDat communication at high clock frequency - Disturbances on the encoder line (e.g. through insufficient shielding) **Error correction** - Check additional information from alarm message 0x1800F - Check the encoder line - Check whether the encoder line is suited for digital transmission at high frequency - Exchange the encoder - Exchange the hardware (CC) - Inform your service agency.
237-1800A	**Error message** 1800A Communic. error, EnDat pos. encoder %1, error code %4 **Cause of error** - An error occurred with the EnDat position encoder - The error code describes the cause: - 101 and 102: The encoder generated an internal error message. A further alarm 0x18010 describes the exact cause. - 103, 104 and 105: An error occurred in communication. Possible causes: - Defective encoder line - Encoder line is not suitable for digital EnDat communication at high clock frequency - Disturbances on the encoder line (e.g. through insufficient shielding) **Error correction** - Check additional information from alarm message 0x18010 - Check the encoder line - Check whether the encoder line is suited for digital transmission at high frequency - Exchange the encoder - Exchange the hardware (CC) - Inform your service agency.

Error number	Description
237-1800B	**Error message** 1800B Axis %1 inverter switch off over -STO.A.x (signal: %4) **Cause of error** - The inverter was switched off by a signal within the control: 1 = "-STO.A.MC.WD" 2 = "-STO.A.P.x" 3 = "-STO.A.PIC" 4 = "-STO.A.CC" **Error correction** - Check the entry in MP_vCtrlTimeSwitchOff (up to NCK version 597110-13) or MP_delayTimeSTOatSS1 (as of NCK version 597110-14) and, if necessary, increase the value in MP_delayTimeSTOatSS1 or reduce the value in MP_vCtrlSwitchOffDelay. - Inform your service agency
237-1800C	**Error message** 1800C Reinitialization of the motor %1 is required **Cause of error** - The drive has to be reinitialization because the max. encoder frequency (VCtrlEncoderInputFunctions, bit 0) was changed **Error correction** - Reinitialize the encoder - Deselect the axis with the machine parameter axisMode (bit x = 0) - Exit the MP editor: - Reactivate the axis in machine parameter axisMode (bit x = 0) and set posEncoderType to the desired value - Re-exit the MP editor - or reset the control (restart) - Inform your service agency
237-1800D	**Error message** 1800D Encoder frequency up to 800 kHz not supported %1 **Cause of error** - The max. encoder frequency of 800 kHz was selected for the motor encoder (VCtrlEncoderInputFunctions, bit 0 = 1) The hardware does not support this frequency. **Error correction** - Set the max. encoder frequency to 500 kHz (VCtrlEncoderInputFunctions, bit 0 = 0) - Inform your service agency

Error number	Description
237-1800E	**Error message** 1800E Excessive motor encoder frequency %1 **Cause of error** - The permissible encoder frequency at the motor encoder input was far exceeded. The monitoring thresholds are: - VCtrlEncoderInputFunctions, bit 0 = 0: 600 kHz - VCtrlEncoderInputFunctions, bit 1 = 1: 1000 kHz **Error correction** - Reduce the max. motor speed - Set the parameter of the motor encoder's input circuit to high frequency (not with CC 424): VCtrlEncoderInputFunctions bit 0 = 1 - Inform your service agency
237-1800F	**Error message** 1800F Error in EnDat motor encoder %1, error code %4 **Cause of error** - The EnDat motor encoder reports an error - The error code describes the cause: 1 = Failure of the illumination 2 = Incorrect signal amplitude 4 = Incorrect position value 8 = Overvoltage in the power supply 16 = Undervoltage in the power supply 32 = Overcurrent 64 = Battery exchange needed **Error correction** - Check the installation of the encoder - Check the power supply of the encoder - If required, exchange the battery - Exchange the encoder - Inform your service agency.

Error number	Description
237-18010	**Error message** 18010 Error in EnDat position encoder %1, error code %4 **Cause of error** - The EnDat position encoder reports an error - The error code describes the cause: 1 = Failure of the illumination 2 = Incorrect signal amplitude 4 = Incorrect position value 8 = Overvoltage in the power supply 16 = Undervoltage in the power supply 32 = Overcurrent 64 = Battery exchange needed **Error correction** - Check the installation of the encoder - Check the power supply of the encoder - If required, exchange the battery - Exchange the encoder - Inform your service agency.
237-18011	**Error message** 18011 Motor encoder: %1 Cause: %4, source: %5, alarm: %6 **Cause of error** - An error occurred during forced dynamic sampling of error statuses in the EnDat 2.2 motor encoder **Error correction** - Check the encoder cable - Check the encoder and, if necessary, exchange it - Check the CC controller unit or UEC and, if necessary, exchange it - Inform your service agency Please indicate the complete error text: cause, source and alarm
237-18012	**Error message** 18012 Position encoder: %1 Cause: %4, source: %5, alarm: %6 **Cause of error** - An error occurred during forced dynamic sampling of error statuses in the EnDat 2.2 position encoder **Error correction** - Check the encoder cable - Check the encoder and, if necessary, exchange it - Check the CC controller unit or UEC and, if necessary, exchange it - Inform your service agency Please indicate the complete error text: cause, source and alarm

Error number	Description
237-18013	**Error message** 18013 DQ encoder %1: error condition %4 determined **Cause of error** - One of the safety signals of the DRIVE-CLiQ encoder reports a device error. - Meaning of the additional information: 101 = Internal error bit F1 is set. 102 = Internal error bit F2 is set. 103 = Internal bit "Position Ok PO" is missing. 104 = Internal error bit "Fault Severity 0 XG1" is set. **Error correction** - Exchange the defective encoder. - Inform your service agency.
237-18014	**Error message** 18014 Sp. head change err. in DQ axis %1, port %4, err. %5 **Cause of error** An error occurred during a spindle head change accompanied by PHY Power Down and Up. The corresponding PHY is assigned to the given port. Error = error code (BMCR = "Basic mode control" register of the PHY): 1: BMCR before PHY Power Down not OK 2: BMCR after PHY Power Down not OK 3: BMCR after PHY Power Up not OK 4: BMCR after PHY Power Up is OK, but timeout **Error correction** Inform your service agency.
237-18015	**Error message** 18015 PWM freq. axis %1 <= 4 kHz. Restart required. **Cause of error** During initial servicing of the control the PWM frequency of an axis is less than or equal to 4 kHz. This axis is not assigned to the I2C master CC. **Error correction** - Switch the control off and on again - Alarm occurs in spite of repeated restarts: - Exchange defective CC or - Exchange the backup battery of the MC - Inform your service agency.

Error number	Description
237-18017	**Error message** 18017 SPI expansion module: transmission error %4 **Cause of error** - The SPI plug-in board (module for CC or MC) for controlling analog axes or for reading analog actual values cannot respond correctly. - SPI plug-in module is defective. - Wiring error on the SPI plug-in module **Error correction** - Check the external wiring, especially analog inputs and outputs connected to the SPI plug-in module. - Exchange the SPI plug-in board (module for CC or MC). - Inform your service agency.
237-18018	**Error message** 18018 SPI expansion module: error in module, number %4 **Cause of error** - The SPI plug-in board (module for CC or MC) for controlling analog axes or for reading analog actual values cannot respond correctly. - SPI plug-in module is defective. - Wiring error on the SPI plug-in module **Error correction** - Check the external wiring, especially analog inputs and outputs connected to the SPI plug-in module. - Exchange the SPI plug-in board (module for CC or MC). - Inform your service agency.
237-18019	**Error message** 18019 Clearable positioning error %1, ES %4 **Cause of error** A clearable positioning error has occurred (following error too large) due to an active emergency-stop input. In this case, the CC controller unit brakes the drive immediately. The result is a following error. The additional info indicates the active emergency-stop input: 1 = Emergency stop A (ES.A) 2 = Emergency stop A with handwheel (ES.A.HW) 3 = Emergency stop B (ES.B) 4 = Emergency stop B with handwheel (ES.B.HW) 5 = Emergency stop B Functional Safety 6 = Emergency stop B Functional Safety with handwheel **Error correction** - Check the external wiring, especially the emergency stop inputs. - Inform your service agency

Error number	Description
237-1801A	**Error message** 1801A Non-clearable positioning error %1, ES %4 **Cause of error** A positioning error that cannot be cleared (excessive servo lag) occurred as the result of an active emergency-stop input. In this case, the CC controller unit brakes the motor immediately. This results in a servo-lag error. The additional information indicates the active emergency-stop input: 1 = Emergency Stop A 2 = Emergency Stop A Handwheel 3 = Emergency Stop B 4 = Emergency Stop B Handwheel 5 = Emergency Stop B Functional Safety 6 = Emergency Stop B Functional Safety Handwheel 7 = Emergency Stop A Functional Safety 8 = Emergency Stop A Functional Safety Handwheel **Error correction** - Check the external wiring, especially the emergency-stop inputs - Inform your service agency
237-1801B	**Error message** 1801B SPI module in wrong slot **Cause of error** A single SPI module (e.g. CMA-H) solely in SPI slot 2 is not permissible. **Error correction** - Insert the SPI module in slot 1 of the controller unit. Only once the control has been switched off and is free of potential! - If the problem continues, inform your service agency.
237-1801C	**Error message** 1801C SPI expansion board not supported **Cause of error** - Hardware (CC controller) does not support the SPI expansion board. - CC controller assembly is too old. **Error correction** - Replace CC controller assembly with a current model. - Inform your service agency.

Error number	Description
237-1801D	**Error message** 1801D Overcurrent at analog output on SPI expansion module **Cause of error** Current at output exceeds 20 mA Possible causes: - Short circuit to 0 V or to other voltage sources - Short circuit to other outputs **Error correction** - Check the connection of the shield (on both sides, low impedance. Refer to your control's Technical Manual). - Check the cabling for short circuits to 0 V, to voltage sources or outputs of other channels. - Check the input impedance of the receiver. - Inform your service agency.
237-1801E	**Error message** 1801E Excessive temperature at SPI expansion module **Cause of error** Temperature in the output driver exceeded 105 °C. **Error correction** Inform your service agency.
237-1801F	**Error message** 1801F Supply voltage on the SPI expansion module too low **Cause of error** Supply voltage of the output driver is too low: - Voltage drop due to overload or short circuit - Supply voltage through CC/UEC/MC too low **Error correction** - Check the supply voltage. - Check the wiring of the outputs. - Avoid overloads by deactivating one or more output channels. - Inform your service agency.
237-18020	**Error message** 18020 Overcurrent on the analog output of the SPI expansion module **Cause of error** Message accompanies the ¿SHORT_CIRCUIT¿, error message if the error occurred only briefly and corrected itself. **Error correction** No remedial actions required because the error has been settled.

Error number	Description
237-18021	**Error message** 18021 Error in the CMP file: Supply pointer (SP) is faulty **Cause of error** CMP file: Supply pointer (SP) "UCCS" compensations is faulty. The supply pointer points to the infeed point for which the respective compensation is active. Possible input values: -1: Void -> Output value of the block can be used for UCCP. 0: UCCS block 0 1: UCCS block 1 2: UCCS block 0 3: UCCS block 3 4: UCCS block 4 5: UCCS block 5 6: IqNom 7: WNom **Error correction** - Deactivate CMP file in machine parameters - Create CMP file again. - Inform your service agency.
237-18022	**Error message** 18022 CMP file: Axis transfer in the "UCCS" compensation is faulty. **Cause of error** CMP file: The axis transfer in the "UCCS" compensations is faulty. The axis information from the MC, i.e. the transferred axis index does not agree with the axis index of the active axis. There is an internal software error. **Error correction** - Deactivate CMP file in machine parameters - Create CMP file again. - Inform your service agency.

Error number	Description
237-18023	**Error message** 18023 CMP file: Def. function (FUNC) in UCCS compensations is faulty **Cause of error** CMP file: A faulty function (FUNC) was used in the UCCS compensations. The following functions are possible: 0: DoNothing (-> deactivation) 1: Polynomial 2: Inverse polynomial 3: IIR filter, 2nd order 4: Sine 5: Hyperbolic sine 6: Segment-defined function 7: Adaptive filter **Error correction** - Deactivate CMP file in machine parameters - Create CMP file again. - Inform your service agency.

Error number	Description
237-18024	**Error message** 18024 CMP file: Signal definition SIG0 or SIG1 of UCCS is faulty **Cause of error** CMP file: Definition of the input signals SIG0 or SIG1 von UCCS/UCCP is faulty. The following input quantities are allowed: –1: Not active 0: Value entered in the machine parameter (only UCCP) 1: Output value for block 0 2: Output value for block 1 3: Output value for block 2 4: Output value for block 3 5: Output value for block 4 6: Feedforward current 7: Feedforward acceleration 8: Feedforward velocity 9: Nominal speed 10: Following error 11: Nominal current 12: Integral current 13: Nominal voltage 14: Ud 15: Uq 16: IqNom 17: IdNom 18: IqAct 19: IdAct 20: Motor temperature 21: Commutation angle **Error correction** - Deactivate CMP file in machine parameters - Create CMP file again. - Inform your service agency.
237-18025	**Error message** 18025 CMP file: Axis index (SAX) in the UCCS compensations is faulty **Cause of error** CMP file: Axis index (SAX) in the UCCS compensations is faulty. The axis index points at its own axis or another axis that must be located on the same CC board. The given index is with respect to the index of the machine-parameter file. **Error correction** - Deactivate CMP file in machine parameters - Create CMP file again. - Inform your service agency.

Error number	Description
237-18026	**Error message** 18026 CMP file: Entry (ENTR) in the "UCCS" compensations is faulty **Cause of error** CMP file: Entry (ENTR) in the "UCCS" compensations is faulty. "Entry" defines whether the UCCS compensation is applied additively or multiplicatively. The following input is allowed: 0: Application deactivated 1: Additive application 2: Multiplicative application **Error correction** - Deactivate CMP file in machine parameters - Create CMP file again. - Inform your service agency.
237-18027	**Error message** 18027 "Expanded compensations" and "TRC" are not simultaneous **Cause of error** The "TRC = Torque Ripple Compensation" and "Expanded compensations" functions could not be used at the same time. **Error correction** - Deselect the TRC compensation through the appropriate machine parameter or - Select the expanded compensations - Inform your service agency
237-18028	**Error message** 18028 CC%2: invalid command %4 received 1803B CC%2: command %4 received with invalid addressing %5 **Cause of error** A communication error occurred between the main computer (MC) and the controller unit (CC). Possible causes: - HSCI cabling is faulty - Internal software error - The installed software version has not been enabled. **Error correction** - Check the HSCI cabling - Check the the HSCI connector (is it correctly plugged in?) - Check the software version. - Inform your service agency.

Error number	Description
237-18029	**Error message** 18029 CC%2 incrmtl. actl. val. measrmt. w/ motor enc. %1 **Cause of error** - Encoder defective - Data transfer from encoder faulty - Internal software error **Error correction** - Check the encoder cable and its connection - Exchange the encoder cable - Exchange the encoder - Check the software version. - Inform your service agency.
237-18030	**Error message** 18030 CC%2 incrmtl. actl. value measrmt. w/ pos. enc. %1 **Cause of error** - Encoder defective - Data transfer from encoder faulty - Internal software error **Error correction** - Check the encoder cable and its connection - Exchange the encoder cable - Exchange the encoder - Check the software version. - Inform your service agency.
237-18031	**Error message** 18031 CC%2 nominal speed value (PWM) incorrect %1 **Cause of error** The controller unit provides no valid nominal speed value for the motor, or provides it too late: - Controller configuration (machine parameters) faulty - Internal software error **Error correction** - Check the controller configuration or machine parameters for PWM frequency, encoder inputs/outputs and nominal value outputs - Check the software version. - Inform your service agency.
237-18032	**Error message** 18032 Maximum number of CCs was exceeded **Cause of error** The permissible number of CC controller units has been exceeded. **Error correction** - Reduce the number of CCs. - Inform your service agency.

Error number	Description
237-18033	**Error message** 18033 Syntax error in PAC compensation file **Cause of error** - Syntax error in the compensation file for PAC - Faulty function type in the compensation file for PAC **Error correction** - Use TNCopt to recreate a compensation file for PAC - Inform your service agency
237-18034	**Error message** 18034 Invalid axis assignment in PAC compensation file **Cause of error** - Invalid axis assignment in the compensation file for PAC - Syntax error in the compensation file for PAC **Error correction** - Check the axis assignment in the compensation file for PAC - Use TNCopt to recreate a compensation file for PAC - Inform your service agency
237-18035	**Error message** 18035 Syntax error in CTC compensation file **Cause of error** - Invalid input point in the compensation file for CTC - Syntax error in the compensation file for CTC **Error correction** - Use TNCopt to recreate a compensation file for CTC - Inform your service agency
237-18036	**Error message** 18036 CC%2 Drive:%1 stop=%4 cause=%5 axis group=%6 condition=%7 **Cause of error** **Error correction**
237-18037	**Error message** 18037 Required software option not enabled: %4 **Cause of error** The entered software option has not been enabled, although you tried to activate a function connected with it. **Error correction** - Check the software options using the SIK code number - Contact a representative of the machine manufacturer or HEIDENHAIN.

Error number	Description
237-18038	**Error message** 18038 Incorrect entry in CTC compensation file **Cause of error** - Syntax error in the compensation files of the “expanded compensation” of the CTC function **Error correction** - Use TNCopt to recreate a compensation file for CTC - Inform your service agency
237-18039	**Error message** 18039 Speed encoder line count of %1 is incorrect. %4 was expected. **Cause of error** Rotary encoders: - Encoder line count in the parameter is not equal to the EnDat line count found - The entry in machine parameter "cfgServoMotor --> motStr" is incorrect - Incorrect entry under STR in motor table - EnDat 2.2 rotational speed encoder without line count information must be defined as STR = 1 or "cfgServoMotor --> motStr=1" Linear encoders: - the grating period in the parameter deviates from the grating period found by EnDat - The entry in machine parameter posEncodeType or posEncoderIncr is faulty - For EnDat 2.2 linear encoders without specified grating period, the value of the measuring step (e.g. 1 nm or 10 nm) must be parameterized **Error correction** Rotary encoders: - Enter the correct line count in machine parameter "cfgServoMotor --> motStr" - Enter the displayed line count under STR in the motor table Linear encoders: - Enter the displayed grating period in machine parameter posEncoderDist or posEncoderIncr Inform your service agency.

Error number	Description
237-1803A	**Error message** 1803A Configuration error %1 posEncoderDist=%4, posEncoderIncr=%5 **Cause of error** Faulty configuration: - The context from the machine parameters CfgAxisHardware->posEncoderDist to CfgAxisHardware->posEncoderIncr does not agree with the values from the EnDat encoder - With EnDat 2.2: See the Technical Manual of the control **Error correction** - Check the entries of the machine parameters CfgAxisHardware->posEncoderDist or CfgAxisHardware->posEncoderInc and replace them with the displayed values - Inform your service agency.
237-1803C	**Error message** 1803C Incorrect parameter entry in CC compensation file under UCCS **Cause of error** - Invalid entry in the compensation files of the expanded compensation (CTC, PAC, LAC, ...) - The installed software version does not support this entry **Error correction** - Check the entries of the compensation files - Inform your service agency
237-1803D	**Error message** 1803D Incorrect entry in PAC compensation file **Cause of error** - Invalid entry in the compensation file for PAC - Installed software version does not support this entry **Error correction** - Check the entries of the compensation file - Inform your service agency
237-1803E	**Error message** 1803E Error in current controller **Cause of error** In spite of the given nominal current, the current controller could not measure any actual current (= 0). Possible causes: - Line to motor interrupted - Error in the control of the wye-delta contactor - Defective power module **Error correction** - Check the motor lines - Check the wiring and function of the wye-delta contactor - Exchange the power module, if necessary - Inform your service agency

Error number	Description
237-1803F	**Error message** 1803F Syntax error in compensation file LAC **Cause of error** Error in the syntax of the "expanded compensations" Type of the "Load Adaptive Control" function is unknown **Error correction** Generation of a new LAC compensation files through TNCopt Inform your service agency.
237-18041	**Error message** 18041 Number of maximum compensation blocks (CTC/PAC) exceeded **Cause of error** – Maximum number of CTC blocks exceeded – Maximum number of PAC blocks exceeded – Maximum number of other blocks of the extended compensations exceeded **Error correction** – Revise extended compensation files with TNCopt – Deactivation of extended compensations via MP2700/CfgControllerComp.enhancedComp - Inform your service agency
237-18042	**Error message** 18042 Syntax error in compensation file MAC **Cause of error** Error in the syntax of the "expanded compensations" Type of the "Motion Adaptive Control" function is unknown **Error correction** - Use TNCopt to generate a new MAC compensation file - Inform your service agency
237-1804B	**Error message** 1804B There is no compensation file for UCCP 18044 There is no compensation file for CTC 18045 There is no compensation file for PAC 18046 There is no compensation file for LAC 18047 There is no compensation file for MAC 18048 There is no compensation file for ACC 1804A There is no compensation file for UCCS **Cause of error**

Error number	Description
237-1804C	**Error message** 1804C EnDat position encoder: Transmission rate axis %1 **Cause of error** - The communication with the encoder is not possible at the velocity required for servo control. **Error correction** - Check the cable and connecting elements of the signal path - Exchange the encoder - Exchange the CC - Inform your service agency
237-1804D	**Error message** 1804D EnDat motor encoder: Transmission rate axis %1 **Cause of error** - The communication with the encoder is not possible at the velocity required for servo control. **Error correction** - Check the cable and connecting elements of the signal path - Exchange the encoder - Exchange the CC - Inform your service agency
237-1804E	**Error message** 1804E SPI trigger card activates %1 trigger rate %4 Hz **Cause of error** - The synchronization of external encoders through an SPI trigger card in the CC was activated with CfgCCAuxil.miscCtrlFunct1, bit 10 **Error correction** - Check the entry in CfgCCAuxil.miscCtrlFunct1, bit 10 - Inform your service agency

Error number	Description
237-1804F	**Error message** 1804F EnDat2.2 configured but not detected %1 info %4 %5 %6 **Cause of error** An EnDat2.2 encoder was configured, but the control could not switch it to the EnDat2.2-Modus mode. Info contains the following three additional information data: 1) 1: This is about the shaft speed encoder 2: This is about the position encoder 2) Connector offset e.g. connector offset = 3 and additional info 1) = 2: Connector X204 is affected 3) EnDat error word **Error correction** - Check the entry in Machine Parameter CfgAxisHardware-posEncoderType: The value CC_EXTERN_ENDAT_2_2 must not be set unless the position encoder is an EnDat 2.2 encoder. - Check the selected motor: The current settings expect a motor with EnDat2.2 encoder. - Check the entry in Machine Parameter CfgServoMotor-motEncType. - Check the status of the EnDat2.2 encoder with the aid of DriveDiag: Check the pending alarms.
237-18050	**Error message** 18050 CC%2 %1 AVD parameter ID=%4 **Cause of error** There is an error in the AVD parameter file: - The file contains invalid parameter data - An incorrect version of TNCopt may have been used to create it **Error correction** - Deactivate AVD in the configuration (enhancedComp/MP2700) - Make a new AVD file with a current version of TNCopt - Check the software version ID=10 Can only be used on hardware with a limited number of axes - Inform your service agency ID=107 Kv factor of position controller does not match AVD parameter PAR7 ID=108 Kp factor of speed controller does not match AVD parameter PAR8 ID=109 Ki factor of speed controller does not match AVD parameter PAR9 ID=205 File version not valid ID=300 Motor overcurrent due to incorrect AVD parameter assignment

Error number	Description
237-18051	**Error message** 18051 Axis %2: Number of the UV is in error **Cause of error** - Number of the UV power supply unit in the machine parameter is faulty **Error correction** - Check the number of the UV power supply unit in the machine parameter - Inform your service agency
237-18052	**Error message** 18052 Axis %2: Faulty entry in ICTRL compensation file **Cause of error** - Invalid entry in the compensation file for ICTRL - Installed software version does not support this entry **Error correction** - Check the entries in the compensation file for ICTRL - Inform your service agency
237-18056	**Error message** 18056 CC %2 axis %1: Filter %4 in the speed control loop is instable **Cause of error** - The filter parameter set is inconsistent. - The filter frequency defined in the machine parameter MP_vCtrlFilterFreqX is too low or too high. **Error correction** - Check the entries in the machine parameters MP_vCtrl-FilterTypeX, MP_vCtrlFilterFreqX, MP_vCtrlFilterDampingX and MP_vCtrlFilterBandWidthX, whereby X stands for the displayed filter number. - Inform your service agency
237-18057	**Error message** 18057 CC %2 axis %1: Filter %4 in the position control loop is instable **Cause of error** - The filter parameter set is inconsistent - The filter frequency defined in the machine parameter MP_vCtrlFilterFreqX is too low or too high **Error correction** - Check the entries in the machine parameters MP_vCtrl-FilterTypeX, MP_vCtrlFilterFreqX, MP_vCtrlFilterDampingX and MP_vCtrlFilterBandWidthX, whereby X stands for the displayed filter number. - Inform your service agency

Error number	Description
237-18058	**Error message** 18058 CC %2 axis %1: IPC is instable %4 **Cause of error** - IPC time constant is too large **Error correction** - Check machine parameters MP_compIpcT1 and MP_compIpcT2 - Inform your service agency
237-18059	**Error message** 18059 CC %2 axis %1: AVD instable %4 1805A CC %2 axis %1: AVD instable %4 **Cause of error** - The adjusted AVD frequency is too low or too high. **Error correction** - Inspect the AVD frequency with a current version of TNCopt - Generation of a new AVD file through TNCopt - Inform your service agency
237-1805B	**Error message** 1805B CC %2 axis %1: AVD filter is instable %4 **Cause of error** - The parameter set of the AVD filter 13 (type 22) is inconsistent - The defined frequency of the AVD filter 13 (type 22) is too high or too low - The adjusted AVD frequency is too high or too low. **Error correction** - Use the current TNCopt version to check the AVD filter 13. - Inspect the AVD frequency using TNCopt - Generation of a new AVD file through TNCopt - Inform your service agency
237-1805C	**Error message** 1805C CC %2 axis %1: Measurement with AVD is instable %4 18063 CC %2 axis %1: ACC instable %4 18064 CC %2 axis %1: ACC instable %4 18065 CC %2 axis %1: ACC instable %4 18066 CC %2 axis %1: ACC instable %4 1806B CC %2 axis %1: Filter is instable %4 **Cause of error** - Internal software error

Error number	Description
237-1805D	**Error message** 1805D Maximum number of function blocks exceeded %1 **Cause of error** - Maximum number of LAC function blocks exceeded - Maximum number of function blocks of the extended compensations exceeded **Error correction** - Revise extended compensation files with TNCopt - Deactivation of extended compensations via <ParameterSet>.CfgControllerComp.enhancedComp - Inform your service agency
237-1805E	**Error message** 1805E Feed-in point in %4 in line %4 is faulty **Cause of error** - Syntax error in the displayed file - The given feed-in point is not supported **Error correction** - Generation of a new compensation file through TNCopt - Deactivation of extended compensations - Inform your service agency
237-1805F	**Error message** 1805F Error in %4 in line %5 **Cause of error** - Syntax error in the displayed file - Signal index (SIGx) in the displayed line is faulty **Error correction** - Use TNCopt to regenerate the displayed file - Deactivation of extended compensations via <ParameterSet>.CfgControllerComp.enhancedComp - Inform your service agency
237-18060	**Error message** 18060 Faulty assignment in %4 **Cause of error** - Input or output cannot be accessed for compensation (e.g. is on another controller board) - Faulty assignment in the displayed file **Error correction** - Use TNCopt to regenerate the compensation file - Compensations by axis coupling possible only for axes on the same controller board - Inform your service agency

Error number	Description
237-18062	**Error message** 18062 CC %2 axis %1: AVD Filter 1%4 in position control loop instable **Cause of error** - The filter parameter set is inconsistent - The filter frequency is defined too high or too low **Error correction** - Use the current TNCopt version to check the AVD filters 11 and 12 - Generation of a new AVD file through TNCopt - Inform your service agency
237-18067	**Error message** 18067 CC %2 axis %1: CPF is instable %4 **Cause of error** - Separation frequency of the Crossover Position Filters (CPF) is too low **Error correction** - Check the separation frequency (MP_compActDampFreq) - Inform your service agency
237-18068	**Error message** 18068 Axis %1: FSC filter instable (%4) 18069 Axis %1: FSC filter instable (%4) **Cause of error** The FSC filter parameters are faulty. **Error correction** - Check the FSC parameters - Use TNCopt to optimize the FSC parameters - Deactivate FSC (set FscAccTolq to 0) - Inform your service agency
237-1806C	**Error message** 1806C "Machine Parameter" in %4 in line %5 is in error **Cause of error** - Syntax error in displayed file. - The given machine parameter is not supported **Error correction** - Generation of a new compensation file through TNCopt - Deactivation of the expanded compensations - Inform your service agency

Error number	Description
237-1806D	**Error message** 1806D Syntax error in FN22 function **Cause of error** - Error in FN22 syntax **Error correction** Inspect the syntax in the FN22 function Deactivate the FN22 function or the superimposed cycle Inform your service agency
237-1806E	**Error message** 1806E Faulty assignment in %4 **Cause of error** - Faulty assignment in the displayed file **Error correction** - Use TNCopt to regenerate the compensation file - Inform your service agency
237-1806F	**Error message** 1806F Cycle 239 has been canceled **Cause of error** Cycle 239 was canceled No mass/mass moment of inertia was estimated **Error correction** Repeat Cycle 239
237-18070	**Error message** 18070 CC%2 %1 Current limit in control loop **Cause of error** The current limitation in the control loop responded The AVD function was deactivated. Possible causes: - Excessive acceleration - Feedforward-control parameter(s) incorrect - Type of AVD damping incorrect - AVD damping factor too high **Error correction** - Check the nominal acceleration - Check the feedforward-control parameter(s) - Check the AVD parameterization - Adapt the parameter(s) if necessary - - Inform your service agency

Error number	Description
237-18071	**Error message** 18071 CC%2 function only available with EnDat encoder %1 **Cause of error** The crossover position filter is possible only with an EnDat encoder. **Error correction** Deselect the crossover position filter Use of an EnDat linear encoder Inform your service agency
237-18072	**Error message** 18072 Spindle overload %1 **Cause of error** - The spindle was briefly overloaded **Error correction** - Reduce the feed rate - Reduce the infeed - Check the cutting data
237-18074	**Error message** 18074 CC%2 %1 measured mass moment of inertia: %4 [kg'*m*m*0.001] **Cause of error** **Error correction**
237-18076	**Error message** 18076 CC%2 %1 acceleration threshold not reached: %4 [percent] **Cause of error** The required axis acceleration was not reached during the weighing run. The mass or mass moment of inertia can be estimated only if a specific minimum acceleration is reached. **Error correction** - Set override potentiometer to 100% - Increase the traverse distance for the weighing run
237-18078	**Error message** 18078 CC %2 axis %1: parameter %4 not supported by the software. **Cause of error** - Machine parameters are no longer supported by the installed NC software version. **Error correction** - Delete the machine parameters or set them to their initial value. - Inform your service agency

Error number	Description
237-18079	**Error message** 18079 CC%2: disturbance in UM communication, error code=%4 **Cause of error** The communication component for the optical fiber connection (HFL) to the inverters reports an error Possible causes: - Hardware defective - Electromagnetic disturbances - Optical fibers (HFL) not connected correctly - Insufficient bend radius of the optical fibers **Error correction** - Create a service file (error code gives information about the exact cause of the error for the diagnosis at HEIDENHAIN) - Inform your service agency - If required, exchange the hardware - Use TNCdiag to check the damping of the fiber-optic connections (HFL). If the damping is too high: - Check the optical fiber connections - Comply with the guidelines in the Technical Manual regarding the cable routing - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections
237-1807A	**Error message** 1807A CC%2: Internal connection fault, Error code=%4 **Cause of error** The communication component of an internal connection is reporting an error. Possible causes: - Electromagnetic interference - Hardware defective **Error correction** - Check the encoder cables, particularly shield connections - Comply with the guidelines in the Technical Manual regarding the cable routing - Exchange the hardware - Create a service file (error code gives information about the exact cause of the error for the diagnosis at HEIDENHAIN) - Inform your service agency

Error number	Description
237-1807B	**Error message** 1807B CC%2: logical disturbance of data reception %1, error code=%4 **Cause of error** The communication component for the optical fiber connection (HFL) to the inverters reports an error Possible causes: - Hardware defective - Electromagnetic disturbances - Optical fibers (HFL) not connected correctly - Insufficient bend radius of the optical fibers **Error correction** - Create a service file (error code gives information about the exact cause of the error for the diagnosis at HEIDENHAIN) - Inform your service agency - If required, exchange the hardware - Use TNCdiag to check the damping of the fiber-optic connections (HFL). If the damping is too high: - Check the optical fiber connections - Comply with the guidelines in the Technical Manual regarding the cable routing - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections
237-1807C	**Error message** 1807C CC%2: Internal port %5 fault, Error code=%4 **Cause of error** The communication component of an internal connection is reporting an error Possible causes: - Electromagnetic interference - Defective hardware **Error correction** - Check the encoder cables, particularly shield connections - Comply with the guidelines in the Technical Manual regarding the cable routing - Exchange the hardware - Create a service file (error code gives information about the exact cause of the error for the diagnosis at HEIDENHAIN) - Inform your service agency

Error number	Description
237-1807D	**Error message** 1807D CC%2: physical disturbance of data reception %1, error code=%4 **Cause of error** The communication component for the optical fiber connection (HFL) to the inverters reports an error. Possible causes: - Hardware defective - Electromagnetic disturbances - Optical fibers (HFL) not connected correctly - Insufficient bend radius of the optical fibers **Error correction** - Use TNCdiag to check the damping of the optical fiber connections (HFL); if the damping is too high: - Check the optical fiber connections - Comply with the guidelines in the Technical Manual regarding the cable routing - If necessary, exchange the hardware - Create a service file (error code gives information about the exact cause of the error for the diagnosis at HEIDENHAIN) - Inform your service agency - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections
237-1807E	**Error message** 1807E CC%s: Internal connection fault, Error code=%4 **Cause of error** The component for internal optical fiber connections reports an error Possible causes: - Electromagnetic disturbances - Hardware defective **Error correction** - Check the encoder cables, particularly shield connections - Refer to the guidelines in the Technical Manual regarding the cable routing - Exchange the hardware - Create a service file (error code information about the exact cause of the error for the diagnosis at HEIDENHAIN) - Contact your service agency

Error number	Description
237-1807F	**Error message** 1807F UM 3xx inverter not found %1 **Cause of error** - No communication with the inverter of the displayed axis - Optical fibers connected incorrectly or not at all - Supply voltage of the inverter is missing - The machine parameters CfgAxisHardware->inverterInter-face, CfgAxisHardware->motorConnector, or CfgAxisHardware->hsciCcIndex are not parameterized correctly. - Inverter defective **Error correction** - Check the cabling - Check the supply voltage of the inverters - Check the entries in the machine parameters - Replace the inverter - Contact your service agency
237-18082	**Error message** 18082 There is no input for position encoder %1 **Cause of error** - A non-existent input was selected for the position encoder (entry in MP_posEncoderInput) **Error correction** - Check machine parameter MP_posEncoderInput - Inform your service agency
237-18083	**Error message** 18083 There is no input for speed encoder %1 **Cause of error** - A non-existent input was selected for the tachometer (entry in MP_speedEncoderInput) **Error correction** - Check machine parameter MP_speedEncoderInput - Inform your service agency
237-18084	**Error message** 18084 No outlet for optical waveguide %1 **Cause of error** - A non-existent output was selected for the fiber-optic cable to the inverter UM (entry in MP_inverterInterface) **Error correction** - Check the machine parameter MP_inverterInterface - Inform your service agency

Error number	Description
237-18085	**Error message** 18085 There is no motor connection on the inverter %1 **Cause of error** - The selected motor connection (entry in MP_motorConnector) is not present on the UM inverter or UEC compact inverter. **Error correction** - Check the machine parameter MP_motorConnector - Inform your service agency
237-18089	**Error message** 18089 WD monitoring has responded %1 (received:%4, expected:%5) **Cause of error** CC controller unit or UEC compact inverter reports that watchdog is no longer updated on UM inverter or UEC compact inverter **Error correction** - Restart the control - Check the machine for correct shield connections and grounding - Check the motor and power cables for correct shield connections and terminal connections - Inform your service agency
237-1808A	**Error message** 1808A Initial communication with inverter failed CC%2 %1 **Cause of error** No communication could be established with the inverter when booting the control. **Error correction** - Check the power supply - Inform your service agency - Replace the inverter
237-1808B	**Error message** 1808B ICTRL_xx.cmp parameter file faulty **Cause of error** - Incorrect entry in parameter file ICTRL_xx.cmp **Error correction** - Generation of a new parameter file via TNCopt - Variable parameter unknown - Variable parameter not permitted for this function - Inform your service agency

Error number	Description
237-1808C	**Error message** 1808C Parameter entered: "%1 WearAdaptFriction = %4" **Cause of error** - Wear parameter(s) defined **Error correction**
237-1808D	**Error message** 1808D Error in the determination of "WearAdaptFriction" **Cause of error** An error occurred while determining "WearAdaptFriction" - The velocity profile is not suitable for determining the wear parameter - The velocity profile is too long; as a result, the data recording is aborted too soon. - The velocity profile is too short; as a result, no constant velocity is reached for measured-value recording. **Error correction** - Adapt the velocity profile for determining the characteristic value for the wear - Inform your service agency
237-1808E	**Error message** 1808E PWM switching not permitted during operation **Cause of error** - The parameter for the PWM frequency was changed during operation. - The initial machine parameters contain a different PWM frequency. **Error correction** - All machine parameter subfiles must have the same PWM settings. - For all subfiles and all axes that are on the same PWM output, change the PWM frequency to the same value. - Inform your service agency.

Error number	Description
237-1808F	**Error message** 1808F CC%2 incompatible motor system %1, X%4 connection **Cause of error** The configured encoder input of the controller unit can evaluate only digital, purely serial measuring systems This input does not support encoders with 1Vpp signals. **Error correction** - If the motor encoder being used is a 1Vpp encoder: use inputs X401 to X406 of this CC and configure them through CfgAxisConfig/MP_speedEncoderInput. - If it is an encoder with purely serial EnDat interface: set the parameter CfgServoMotor/MP_motEncType to the correct value. - Inform your service agency.
237-18090	**Error message** 18090 CC%2 incompatible position encoder %1, X%4 connection **Cause of error** The configured encoder input of the controller unit can evaluate only digital, purely serial measuring systems This input does not support encoders with 1Vpp signals. **Error correction** - If the position encoder being used is a 1Vpp encoder: use inputs X401 to X406 of the CC and configure them through CfgAxisConfig/MP_posEncoderInput. - If it is an encoder with purely serial EnDat interface: set the parameter CfgAxisHardware/MP_posEncoderType to the correct value. - Inform your service agency
237-18091	**Error message** 18091 Speed controller: P factor or reset time too small in axis %1 **Cause of error** - P-factor of the speed controller in MP_vCtrlPropGain under CfgSpeedControl is too small - Reset time of the speed controller MP_vCtrlPropGain or MP_vCtrlIntGain is too short - Bit 20 of MP_miscCtrlFunct0 under CfgCCAuxil is set, although P factor and reset time are not too small **Error correction** - Increase the P factor in MP_vCtrlPropGain - Reduce the I factor in MP_vCtrlIntGain - Reset bit 20 of MP_miscCtrlFunct0

Error number	Description
237-18094	**Error message** 18094 CC%2 error during EnDat3 initialization %1 **Cause of error** - A problem occurred during initialization of the EnDat3 interface (AddInfo[0] == 0). - Communication with the EnDat3 encoder is not possible (AddInfo[0] == 1). **Error correction** - Restart the control - Exchange the CC controller unit - Check the encoder connecting cable - Remove the encoder from power for at least 5 seconds (disconnect the encoder from the control) and then restart the control - Exchange the encoder
237-18096	**Error message** 18096 Parameters of current controller are not correct %1 info %4 **Cause of error** - The current controller (CfgCurrentControl) is not parameterized correctly Info = 1: Mixed parameterization is not permitted: Please use only (iCtrlPropGain and iCtrlIntGain) or (iCtrlPropGainD, iCtrlIntGainD, iCtrlPropGainQ, and iCtrlIntGainQ) Info = 2: iCtrlPropGain = 0, even though iCtrlIntGain > 0 Info = 3: iCtrlPropGainD = 0, even though iCtrlIntGainD > 0 Info = 4: iCtrlPropGainQ = 0, even though iCtrlIntGainQ > 0 **Error correction** - Correct the parameterization of the current controller Info = 1: Set (iCtrlPropGain = 0 and iCtrlIntGain = 0) or (iCtrlPropGainD = 0, iCtrlIntGainD = 0, iCtrlPropGainQ = 0, and iCtrlIntGainQ = 0) Info = 2: Set iCtrlPropGain > 0 or iCtrlIntGain = 0 Info = 3: Set iCtrlPropGainD > 0 or iCtrlIntGainD = 0 Info = 4: Set iCtrlPropGainQ > 0 or iCtrlIntGainQ = 0
237-18097	**Error message** 18097 Axis %1: measurement not possible **Cause of error** The axis was moved beyond the traverse limit during the measurement. **Error correction** Position the axis so that there is sufficient clearance to the traverse limits. Reduce the underlying velocity.

Error number	Description
237-18098	**Error message** 18098 Axis %1: Application of the excitation signal canceled. **Cause of error** Drive is off or was switched off while the excitation signal was applied. **Error correction** Repeat the measurement with all drives switched on.
237-18099	**Error message** 18099 Shaft-speed-dependent PWM frequency switching not possible %1 **Cause of error** - A shaft-speed-dependent switching of the PWM frequency is configured with CfgCurrentControl/MP_iCtrlPwmType = 2. The software version being used does not yet support this functionality for Gen 3 drives. **Error correction** - Set the parameter CfgCurrentControl->iCtrlPwmType to 0. - If necessary, permanently set the PWM frequency to the higher value. - Inform your service agency.
237-1809A	**Error message** 1809A TNCopt adjustment sequence was started **Cause of error** An automatic adjustment sequence was started in TNCopt. TNCopt is controlling the control. The movement of the axes can be started automatically. **Error correction** - Check whether TNCopt is connected to the control and an adjustment sequence was started.
237-1809B	**Error message** 1809B Axes %1: Oscillation detected. **Cause of error** An oscillation was detected during the measurement. **Error correction** Reduce the amplitude of the excitation signal. Ensure that the control loops have sufficient stability reserves.

Error number	Description
237-1809C	**Error message** 1809C Type of referencing not permissible %1 **Cause of error** The mode set in the parameter CfgReferencing->refType is not possible with the present encoder. **Error correction** Enter a referencing mode suitable to the encoder in the stated parameter.
237-1809D	**Error message** 1809D CC%2: faulty internal connection **Cause of error** Interrupted communication to a device-internal controller. Possible causes: - Supply voltage outside of the permissible tolerance - Hardware defective **Error correction** - Check the supply voltage of the affected CC and dimension it according to the manual. - Exchange the CC. - Inform your service agency.
237-1809E	**Error message** 1809E CC%2: faulty internal connection **Cause of error** Interrupted communication to a device-internal controller. Possible causes: - Supply voltage outside of the permissible tolerance - Hardware defective **Error correction** - Check the supply voltage of the affected CC and dimension it according to the manual. - Exchange the CC. - Inform your service agency.
237-1809F	**Error message** 1809F Axis %1: value of amplitude not in permissible range **Cause of error** The value of the amplitude of the excitation signal for the measurement of the frequency response is outside of the permissible range. **Error correction** - Inform your machine tool builder - Check the OEM macro for Cycle 238 (Measure machine status, CfgSystemCycle->OEM_MACHSTAT_MEAS) - Check the value for the excitation amplitude - Recommended setting: Enter the amplitude as a factor of the rated current. A typical value is 0.3.

Error number	Description
237-180A0	**Error message** 180A0 Non-released inverter software loaded **Cause of error** The inverter contains test software that has not been released. This software may only be used for internal testing purposes. **Error correction** - Check the software version - Generate service files - Inform your service agency
237-180A1	**Error message** 180A1 "Time exceeded" trigger **Cause of error** The event that triggers the trigger did not occur in time. The programmed feed rate was not reached. **Error correction** Reduce the programmed feed rate.
237-180A2	**Error message** 180A2 Axis %1: %4 is not supported by the CC controller unit **Cause of error** The configured function is not supported in combination with the controller unit being used (CC or UxC). **Error correction** - Deactivate the function - Inform your service agency
237-180A3	**Error message** 180A3 Axis %1: zero crossover of the %4 speed in '%5' measurement **Cause of error** While recording the Bode plot, a zero crossover of a measured velocity was detected. For this type of measurement, non-linear effects of friction at velocity=0 should be avoided. **Error correction** Increase the ratio of the underlying velocity to the excitation amplitude

Error number	Description
237-180A4	**Error message** 180A4 CC%2: maximum computing load has risen for axis %1 **Cause of error** Due to a changed or newly activated set of machine parameters, the computing load of the CC has increased compared to the load through the initial or previous set of machine parameters. Factors in the computing load are the parameters for the PWM frequency (CfgPowerStage > ampPwmFreq), the controller performance (CfgAxisHardware > ctrlPerformance), and the current controller cycle time (CfgCurrentControl > iCtrlPwmType). **Error correction** - Reboot the control to initially load the changed set of parameters while booting. - Initially load the set of machine parameters with the highest computing load (if necessary with the attribute Axes->PhysicalAxes->(axis designation)->deactivatedAtStart = TRUE) and only then activate the set of machine parameters appropriate for the current hardware configuration (with less computing load).
237-180A6	**Error message** 180A6 Axis %1: waterfall diagram measurement: v_nom not constant **Cause of error** The nominal velocity v_nom is not constant while recording the waterfall chart. **Error correction** - Inform your machine manufacturer - Check/correct the OEM macro for Cycle 238 Measure Machine Status (CfgSystemCycle > OEM_MACHSTAT_MEAS) - Increase the lead and lag time

Error number	Description
237-180A7	**Error message** 180A7 CC%2: communication error w/ EnDat3 speed encoder, error code %6 180A8 CC%2: communication error w/ EnDat3 pos. encoder, error code %6 **Cause of error** - An error occurred during communication with the EnDat3 encoder - The error code describes the cause: - 0, 1: Error in the physical layer (PHY) - 2: CRC error of the HPF or an LPF - 3: WD error; no response from the encoder was detected during the timeout - Possible causes: - Defective encoder - Defective encoder connecting cable - Noise on the encoder connecting cable **Error correction** - Exchange the encoder - Check the connecting cable, and exchange it if necessary - Exchange the CC controller unit
237-180A9	**Error message** 180A9 CC %2: EnDat3 speed encoder reports error, error code %6 180AA CC %2: EnDat3 position encoder reports error, error code %6 **Cause of error** The EnDat3 encoder reports an error during cyclic communication The error code describes the cause: - 6: Generation of data failed or encoder system failure - 7: Singleturn position faulty - 8: Multiturn position faulty - 9: Permissible ambient conditions of the encoder exceeded (e.g. temperature) - 10: Permissible electrical operating conditions exceeded (current and/or voltage too high or too low) - 11, 12, 13, 14: Device-specific error message 0-3 (note the encoder documentation) - 15: Unspecified error message **Error correction** - Check the encoder - Ensure the correct ambient conditions of the encoder - Exchange the encoder

Error number	Description
237-180AB	**Error message** 180AB CC%2: error in cyclic com. w/ EnDat3 speed encoder, error code %6 180AC CC%2: error in cyclic com. w/ EnDat3 pos. encoder, error code %6 **Cause of error** The encoder reports an error during cyclic communication The error code describes the cause: - 4: Invalid position data were transmitted - 5: A request code is not supported by the encoder - 6: An error that cannot be associated was signaled **Error correction** - Check the encoder - Check any additional queries - Exchange the encoder - Exchange the CC controller unit - Inform your service agency
237-180AD	**Error message** 180AD CC%2: warning by EnDat3 encoder to %10; warning code: %6 **Cause of error** A warning was signaled during communication with the encoder The warning code describes the cause: - 0: Encoder status/maintenance - 1: Close to the limits of the permissible ambient conditions (e.g. temperature) - 2: Close to the limits of the permissible electrical operating conditions (current/voltage) - 3: Temperature warning threshold exceeded - 4: Almost down to minimum battery charge - 5-8: Encoder-specific warning 0-3 - 9: Unspecified warning from encoder - 10: A warning that cannot be associated is being signaled A serious encoder error could occur soon **Error correction** Measures depending on the warning code: - 0: Servicing of the encoder is recommended - 1, 4: Check compliance with the ambient conditions (e.g. temperature) and initiate measures - 2: Check compliance with the electrical operating conditions (e.g. ensure the voltage supply) - 3: Ensure compliance with the electrical operating conditions, change the battery soon - 5-8: Initiate measures as per the encoder documentation - 9, 10: Check the encoder; exchange the encoder; inform your service agency

Error number	Description
237-180B1	**Error message** 180B1 Time exceeded during communication with UM 3xx at %1 **Cause of error** The described UM 3xx did not react to a communication request from the CC in time. Possible causes - Electromagnetic noise or faulty electrical contacts of the 24V supply voltage - Electromagnetic noise or faulty electrical contacts of the DC-link voltage - Electromagnetic noise or faulty electrical contacts in the motor cabling - Electromagnetic noise or faulty electrical contacts in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine and wiring for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing, and also for contamination and correct clamping - Exchange the UM 3xx inverter - Exchange the CC 3xx controller unit - Generate a service file and inform your service agency
237-180B2	**Error message** 180B2 Error in communication with UM 3xx at %1 **Cause of error** Telegrams were lost during transmission of the actual current values from the UM to the CC. Possible causes - Electromagnetic noise or faulty electrical contacts of the 24V supply voltage - Electromagnetic noise or faulty electrical contacts of the DC-link voltage - Electromagnetic noise or faulty electrical contacts in the motor cabling - Electromagnetic noise or faulty electrical contacts in the brake cabling - Contamination or poor optical coupling of the HFL- - Internal system error **Error correction** - Check the machine and wiring for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing, and also for contamination and correct clamping - Exchange the UM 3xx inverter - Exchange the CC 3xx controller unit - Generate a service file and inform your service agency

Error number	Description
237-180B3	**Error message** 180B3 Time exceeded during communication with internal component at %1 **Cause of error** The position-value converter component assigned to the connector did not react to a communication request from the processor in time. Possible causes - Electromagnetic noise or faulty electrical contacts of the 24V supply voltage - Electromagnetic noise or faulty electrical contacts in the encoder cabling - Electromagnetic noise or faulty electrical contacts in the shield connection - Internal system error **Error correction** - Check the machine and wiring for correct shield connection and grounding - Check the encoder cabling for correct clamping - Exchange the affected encoder or other encoders in the same column - so for X401 also X402, X431, or X432 - so for X403 also X404, X433, or X434 - so for X405 also X406, X435, or X436 - Exchange the CC 3xx controller unit - Generate a service file and inform your service agency
237-180B4	**Error message** 180B4 Error in communication with internal component at %1 **Cause of error** Communication with the position-value converter component for the described connector is impaired. Possible causes - Electromagnetic noise or faulty electrical contacts of the 24V supply voltage - Electromagnetic noise or faulty electrical contacts in the encoder cabling - Electromagnetic noise or faulty electrical contacts in the shield connection - Internal system error **Error correction** - Check the machine and wiring for correct shield connection and grounding - Check the encoder cabling for correct clamping - Exchange the affected encoder or other encoders in the same column - so for X401 also X402, X431, or X432 - so for X403 also X404, X433, or X434 - so for X405 also X406, X435, or X436 - Exchange the CC 3xx controller unit - Generate a service file and inform your service agency

Error number	Description
237-180B5	**Error message** 180B5 Too many faulty telegrams of a UM 3xx in sequence %1 **Cause of error** Too many telegrams in sequence were lost during transmission of the actual current values from the UM to the CC. Possible causes - Electromagnetic noise or faulty electrical contacts of the 24V supply voltage - Electromagnetic noise or faulty electrical contacts of the DC-link voltage - Electromagnetic noise or faulty electrical contacts in the motor cabling - Electromagnetic noise or faulty electrical contacts in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine and wiring for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing, and also for contamination and correct clamping - Exchange the UM 3xx inverter - Exchange the CC 3xx controller unit - Generate a service file and inform your service agency
237-180B6	**Error message** 180B6 Too many faulty telegrams of a UM 3xx %1 **Cause of error** Too many telegrams were lost during transmission of the actual current values from the UM to the CC during the monitoring period. Possible causes - Electromagnetic noise or faulty electrical contacts of the 24V supply voltage - Electromagnetic noise or faulty electrical contacts of the DC-link voltage - Electromagnetic noise or faulty electrical contacts in the motor cabling - Electromagnetic noise or faulty electrical contacts in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine and wiring for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing, and also for contamination and correct clamping - Exchange the UM 3xx inverter - Exchange the CC 3xx controller unit - Generate a service file and inform your service agency

Error number	Description
237-180B7	**Error message** 180B7 Excessive error frequency for telegrams of UM 3xx %1 **Cause of error** The frequency of faulty telegrams during transmission of the actual current values from the UM to the CC is above the limit value. Possible causes - Electromagnetic noise or faulty electrical contacts of the 24V supply voltage - Electromagnetic noise or faulty electrical contacts of the DC-link voltage - Electromagnetic noise or faulty electrical contacts in the motor cabling - Electromagnetic noise or faulty electrical contacts in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - Check the machine and wiring for correct shield connection and grounding - Check the power cables for correct clamping - Check the HFL for correct routing, and also for contamination and correct clamping - Exchange the UM 3xx inverter - Exchange the CC 3xx controller unit - Generate a service file and inform your service agency
237-180B8	**Error message** 180B8 Conspicuous telegrams of UM 3xx %1 below the reporting threshold **Cause of error** There are peculiar telegrams during transmission of the actual current values from the UM to the CC. Possible causes - Electromagnetic noise or faulty electrical contacts of the 24V supply voltage - Electromagnetic noise or faulty electrical contacts of the DC-link voltage - Electromagnetic noise or faulty electrical contacts in the motor cabling - Electromagnetic noise or faulty electrical contacts in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - No immediate corrective action is necessary, since no error situation has occurred yet - Possible preventive measure: check the machine for correct shield connection and grounding - Possible preventive measure: check the power cables for correct clamping - Possible preventive measure: check the HFL for correct routing, and also for contamination and correct clamping

Error number	Description
237-180B9	**Error message** 180B9 Conspicuous telegrams of UM 3xx %1 **Cause of error** There are peculiar telegrams during transmission of the actual current values from the UM to the CC. Possible causes - Electromagnetic noise or faulty electrical contacts of the 24V supply voltage - Electromagnetic noise or faulty electrical contacts of the DC-link voltage - Electromagnetic noise or faulty electrical contacts in the motor cabling - Electromagnetic noise or faulty electrical contacts in the brake cabling - Contamination or poor optical coupling of the HFL **Error correction** - No immediate corrective action is necessary, since no error situation has occurred yet - Possible preventive measure: check the machine for correct shield connection and grounding - Possible preventive measure: check the power cables for correct clamping - Possible preventive measure: check the HFL for correct routing, and also for contamination and correct clamping
237-180BA	**Error message** 180BA CC%2 EnDat2.2 incremental: faulty absolute value %1 **Cause of error** - Reference run was faulty - Reversal during the reference run - The same reference mark was traversed more than once **Error correction** - Separate the encoder from the power supply (for at least 10 seconds) - Switch off the power supply of the machine (main power switch off and on)
237-180BB	**Error message** 180BB Faulty FSuC Include file CC%2 %1 **Cause of error** The interface versions of internal components (SOC and FSuC) on the CC do not match. **Error correction** - Check the software version and run an update if necessary - Inform your service agency

Error number	Description
237-180BC	**Error message** 180BC Field-angle adjustment: chkPosHoldFieldAdj = 0 axis %1 **Cause of error** - A field angle adjustment was started even though the parameter CfgControllerAuxil > chkPosHoldFieldAdj has the value 0 - Only for spindles is it possible to deactivate position monitoring during a field angle adjustment - This is not permitted for axes **Error correction** - Set CfgControllerAuxil > chkPosHoldFieldAdj to a value greater than 0
237-180BD	**Error message** 180BD Position monitoring for field-angle adjustment %1, ES %4 **Cause of error** - The maximum position deviation was exceeded during the field angle adjustment. The additional information indicates the active emergency-stop input, if set: 0 = No emergency-stop input is set 1 = Emergency Stop A 2 = Emergency Stop A Handwheel 3 = Emergency Stop B 4 = Emergency Stop B Handwheel 5 = Emergency Stop B Functional Safety 6 = Emergency Stop B Functional Safety Handwheel 7 = Emergency Stop A Functional Safety 8 = Emergency Stop A Functional Safety Handwheel **Error correction** - Check the parameter CfgControllerAuxil > chkPosHoldField-Adj and increase it, if necessary - Check the external wiring, especially the emergency-stop inputs - Check the encoder and motor data
237-180BE	**Error message** 180BE Monitoring of servo lag inactive %1 **Cause of error** - The parameter CfgPosControl > servoLagMax2 has the value 0 - Only for spindles is it possible to deactivate servo-lag monitoring. This is not permitted for axes. **Error correction** - Set CfgPosControl > servoLagMax2 to a value greater than 0

Error number	Description
237-18500	**Error message** 18500 SKERN-CC%2: encoder error CRC X%4 **Cause of error** - Checksum error for the encoder data **Error correction** - Reboot the control - Exchange the controller unit (CC or UEC) if necessary - Inform your service agency
237-18501	**Error message** 18501 SKERN-CC%2: encoder error BlockID X%4 **Cause of error** - Block ID error for the encoder data **Error correction** - Reboot the control - Exchange the controller unit (CC or UEC) if necessary - Inform your service agency
237-18502	**Error message** 18502 SKERN-CC%2: encoder error amplitude too small X%4 18800 SKERN-CC%2: encoder warning amplitude too small X%4 **Cause of error** - Amplitude of encoder is too low **Error correction** - Check power supply of the encoder - Check encoder cabling - Replace encoder - Inform your service agency
237-18503	**Error message** 18503 SKERN-CC%2: encoder error amplitude too large X%4 **Cause of error** - Amplitude of encoder is too high **Error correction** - Check encoder cabling - Replace encoder if necessary - Inform your service agency
237-18504	**Error message** 18504 SKERN-CC%2: encoder error frequency incorrect X%4 **Cause of error** - Frequency error in the encoder signals **Error correction** - Check encoder cabling including the shielding - Replace encoder if necessary - Inform your service agency

Error number	Description
237-18505	**Error message** 18505 SKERN-CC%2: encoder error latch counter not incremented X%4 **Cause of error** - Faulty latch counter in the encoder data - No new encoder data received **Error correction** - Reboot the control - Exchange the controller unit (CC or UEC) or encoder (only if EnDat) if necessary - Check the encoder connection and cabling - Inform your service agency
237-18506	**Error message** 18506 SKERN-CC%2: encoder error pin ID X%4 **Cause of error** - PIN ID in the encoder data incorrect **Error correction** - Exchange the controller unit (CC or UEC) if necessary - Inform your service agency
237-18507	**Error message** 18507 SKERN-CC%2: encoder error EnDat22 F1 bit set X%4 **Cause of error** - Error bit F1 set in the EnDat22 encoder **Error correction** - Pay attention to the subsequent alarms - Check/replace encoder cabling - Replace encoder - Inform your service agency
237-18508	**Error message** 18508 SKERN-CC%2: encoder error EnDat22 F2 bit set X%4 **Cause of error** - Error bit F2 set in the EnDat22 encoder **Error correction** - Pay attention to the subsequent alarms - Check/replace encoder cabling - Replace encoder - Inform your service agency

Error number	Description
237-18509	**Error message** 18509 SKERN-CC%2: encoder error EnDat22 CRC X%4 **Cause of error** - Checksum error for the EnDat22 encoder data **Error correction** - Check/replace encoder cabling - Replace encoder - Inform your service agency
237-18510	**Error message** 18510 SKERN-CC%2: encoder error invalid overflow value X %4 **Cause of error** - Invalid overflow value for the encoder data **Error correction** - Inform your service agency
237-18511	**Error message** 18511 SKERN-CC%2: encoder error: group error in the EnDat master X%4 **Cause of error** There is an error with the encoder (EnDat master). **Error correction** Inform your service agency
237-18520	**Error message** 18520 SKERN-CC%2: CRC error in cyclic inverter communication %1 18523 UM (FS.B): CRC error during cyclic UM communication CC%2 %1 **Cause of error** - CRC error in cyclic FS communication with the inverter **Error correction** - Reboot the control - Inform your service agency
237-18521	**Error message** 18521 SKERN-CC%2: counter error in cyclic inverter communication %1 18524 UM (FS.B): counter error during cyclic UM communication CC%2 %1 **Cause of error** - Faulty packet counter in cyclic FS communication with the inverter **Error correction** - Reboot the control - Inform your service agency

Error number	Description
237-18522	**Error message** 18522 SKERN-CC%2: checksum faulty %1 18525 UM (FS.B): checksum error CC%2 %1 **Cause of error** - Checksum error of the UM-DriveID in cyclic FS communication with the inverter **Error correction** - Reboot the control - Inform your service agency
237-18530	**Error message** 18530 SKERN-CC%2: axis state during SMP change not STO %1 **Cause of error** - The axis was not in STO when one of the following parameters was changed: hsciCcIndex, inverterInterface, motorConnector **Error correction** - Switch the affected drive off or put it in STO before changing the parameter(s) - Check the PLC/SPLC program, and adapt it if necessary - Inform your service agency
237-18531	**Error message** 18531 SKERN-CC%2: watchdog error SKERN B400 SKERN-CC%2: watchdog error interface **Cause of error** - Internal software error (watchdog low priority cycle) **Error correction** - Inform your service agency
237-18540	**Error message** 18540 Encoder at X%4: Error during test of signal amplitude **Cause of error** An encoder with incremental signals is connected to the indicated connection. An error occurred with this encoder while testing the signal amplitude: - The dynamic sampling of an excessively high signal amplitude could not be completed successfully - The dynamic sampling of an excessively low signal amplitude could not be completed successfully **Error correction** - Check connections and cable of the encoder - Replace encoder or cable - Inform your service agency

Error number	Description
237-18541	**Error message** 18541 EnDat encoder at X%4: Forced dynamic sampling failed **Cause of error** An encoder with EnDat interface is connected to the indicated connection. An error occurred with this encoder during forced dynamic sampling. Possible causes: - Encoder not connected correctly - Encoder defective **Error correction** - Check connections and cable of the encoder - Replace encoder or cable - Inform your service agency
237-18542	**Error message** 18542 SKERN-CC%2: single-event-upset error (SEU) was determined SKERN-MC: single-event-upset error (SEU) was determined **Cause of error** - Internal software error - Possible sporadic error due to EMC irradiation **Error correction** - Restart the control - Check the shielding or shield connection of the devices - Shield or remove possible EMC interferences - Inform your service agency
237-18544	**Error message** 18544 SKERN-CC%2: Inverter SS0 request axis %1 18554 UM (FS.B): CC%2 SS0 request %1 **Cause of error** - The inverter requests an SS0 stop function through the cyclic UM(FS.B) communication **Error correction** - Note any further messages from the inverter - Reboot the control
237-18545	**Error message** 18545 SKERN-CC%2: Inverter SS1F request axis %1 18555 UM (FS-B): UM requests SS1F CC%2 %1 **Cause of error** - The inverter requests an SS1F stop function through the cyclic UM(FS.B) communication **Error correction** - Note any further messages from the inverter - Reboot the control

Error number	Description
237-18546	**Error message** 18546 SKERN-CC%2: Inverter watchdog error FSuC axis %1 18556 UM (FS.B): watchdog error UM-FS.A CC%2 %1 **Cause of error** - FS.B of the inverter reports a watchdog error FS.A (WDF.A) - FS.A of the inverter is no longer ready **Error correction** - Reboot the control - Inform your service agency - Exchange the inverter, if necessary
237-18547	**Error message** 18547 SKERN-CC%2: Inverter FS.A supply voltage faulty axis %1 18557 UM (FS.B): UM-FS.A improper power supply CC%2 %1 **Cause of error** - An internal supply voltage is too high or too low **Error correction** - Note any further messages from the inverter - Check the supply voltage of the inverter (jumper plug X76) - Reboot the control - Inform your service agency - Exchange the inverter, if necessary
237-18548	**Error message** 18548 SKERN-CC%2: Inverter requests an SS1 reaction axis %1 18558 UM (FS.B): UM requests SS1 reaction CC%2 %1 **Cause of error** - The inverter requests an SS1 stop reaction through the cyclic UM(FS.B) communication **Error correction** - Note any further messages from the inverter
237-18549	**Error message** 18549 SKERN-CC%2: Inverter requests an SS2 reaction axis %1 18559 UM (FS.B): UM requests SS2 reaction CC%2 %1 **Cause of error** - The inverter requests an SS2 stop reaction through the cyclic UM(FS.B) communication **Error correction** - Note any further messages from the inverter

Error number	Description
237-18550	**Error message** 18550 Test command was received in released software! 13F01 Test command was received in released software! **Cause of error** Error injection was demanded for a release software. This is not permissible! **Error correction** - Use autotest software! - Inform your service agency
237-18552	**Error message** 18552 SKERN-CC%2: Stuck-At error on the temperature channel **Cause of error** - The A/D converter channel (on the controller unit) for measuring the board temperature is defective - The firmware of the controller unit has detected a fault **Error correction** - Exchange the controller unit (CC or UEC) if necessary - Inform your service agency
237-18553	**Error message** 18553 SKERN-CC%2: Stuck-at error in voltage channel %4 **Cause of error** - Controller unit is defective (ADC channel). - Driver problem of the firmware of the controller unit (CC, UEC). **Error correction** - Inform your service agency. - Exchange the controller unit (CC, UEC), if necessary.
237-1855A	**Error message** 1855A SKERN-CC%2: EnDat forced dynamic sampling not performed **Cause of error** Checking of the Endat22 forced dynamic sampling at the end of the first self-test failed. No successful EnDat forced dynamic sampling performed in the last 168 hours was detected. **Error correction** - Check the encoder for a defect or fault - Exchange the encoder

Error number	Description
237-1855B	**Error message** 1855B SKERN-CC%2: interval violation upon EnDat forced dynamic sampling **Cause of error** - The minimum time interval (4 hours) until the next EnDat forced dynamic sampling was violated (AddInfo[4] = 2) - The maximum time interval (168 hours) until the next EnDat forced dynamic sampling was violated (AddInfo[4] = 1) **Error correction** - Restart the control - If the error recurs, inform your service agency
237-1855C	**Error message** 1855C SKERN-CC%2: errors during EnDat forced dynamic sampling **Cause of error** The detected dynamic error bit does not match the forced dynamic sampling error. **Error correction** - Restart the control - - Check the encoder for a defect or fault - Exchange the encoder - If the error recurs after exchanging the encoder, inform your service agency
237-18801	**Error message** 18801 Autotest manipulation via hcS_AUTO_TEST_SK-ERN_CC is active! **Cause of error** - An error injection was triggered - No officially released software version is installed **Error correction** - Check the software version of the control - Install a released software version - Inform your service agency

Error number	Description
237-18802	**Error message** 18802 SKERN-CC: Test software without safety approval is loaded **Cause of error** The CC contains a test software with safety approval - This software has neither been tested nor released - No checksum will be calculated **Error correction** - After acknowledging the error message you can use this software only for tests - Check the software version - Create service files - Contact your service agency
237-18803	**Error message** 18803 CC%2 synchronization of SPLC run-time system lost **Cause of error** The mechanism for the synchronization of the SPLC run-time system routine on all CC controller units returned an error. **Error correction** - Generate a service file - Contact your service agency - Restart the control

Error number	Description
237-19000	**Error message** 19000 DQ-LT %1: Overcurrent ID=%4; fault value=%5 **Cause of error** The power module of the encoder detected overcurrent. - Closed-loop control is incorrectly parameterized - Motor has a short circuit or ground fault - Volts-per-hertz mode: Run-up ramp is set too small - U/f mode: Rated current of the motor is significantly greater than that of the motor module - Incoming power: High discharging and recharging currents in the event of line power failure - Incoming power: High recharging currents in the event of motor overload and failure of the DC-link voltage - Incoming power: Short circuit currents during switch-on due to missing commutating reactor. - Power lines are not correctly connected. - The power lines exceed the maximum permissible length. - Power module defective. Fault value (interpret bit-wise): Bit 0: Phase U. Bit 1: Phase V. Bit 2: Phase W. **Error correction** - Check the motor data; if required, repeat initial servicing - Check the motor circuit configuration (wye-delta). - Volts-per-hertz mode: Increase the run-up ramp - U/f mode: Check the assignment of the rated currents of the motor and motor modules - Incoming power: Check the line power quality - Incoming power: Decrease the motor load - Incoming power: Correct connection of the line commutating reactor - Check the power cable connections. - Check the power cables for short-circuit or ground fault. - Check the length of the power cable connections. - Exchange the power module

Error number	Description
237-19001	**Error message** 19001 DQ-MotEnc %1: ref. mark distance ID=%4; fault value= %5 **Cause of error** The measured reference-mark distance does not correspond to the parameterized reference-mark distance. For distance-coded encoders, the reference-mark distance is determined from detected pairs of reference marks. This means that if a reference mark is missing, depending on the pair generation, this cannot result in a fault and also has no effect in the system. Fault value (interpret decimal value): Last measured reference-mark distance in increments (4 increments = 1 encoder pulse). The algebraic sign designates the direction of motion when detecting the reference-mark distance. **Error correction** - Check that the encoder cables are routed in compliance with EMC. - Check the plug connections. - Check the encoder type (encoder with equidistant reference marks). - Replace the encoder or encoder cable.
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error. An error occurs in the pp_GenFB_NCC signal (PL system module). The control does not finish a test step correctly **Error correction** - Check whether a previous system error of the control led to the cancellation of a test section - Check the wiring of the signal pp_GenFB_NCC - Inform your service agency

Error number	Description
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error. An error occurs in the –ES.B signal (PL system module). The control does not finish a test step correctly. **Error correction** Check whether a previous system error of the control led to the cancellation of a test section Check the wiring of the signal –ES.B. Inform your service agency
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error. An error occurs in the CVO signal (PL system module). The control does not finish a test step correctly. **Error correction** Check whether a previous system error of the NC led to the cancellation of a test section Check the wiring of the signal CVO Inform your service agency
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error. An error occurs in the RDY.PS (X69-17a) signal of the supply module. The control does not finish a test step correctly. **Error correction** Check whether a previous system error of the NC led to the cancellation of a test section Check the wiring of the signal RDY.PS (X69-17a) Inform your service agency

Error number	Description
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error. An error occurs in the pp_GenMKG signal (SMOP). The control does not finish a test step correctly. **Error correction** Check whether a previous system error of the NC led to the cancellation of a test section Check the wiring of the signal pp_GenMKG Inform your service agency
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error. An error occurs in the pp_AxGrpStateReq == S_STATE_AUTO signal. The control does not finish a test step correctly. **Error correction** Check whether a previous system error of the NC led to the cancellation of a test section Check the wiring of the guard door/signal pp_AxGrpStateReq == S_STATE_AUTO Inform your service agency
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error. An error occurs in the pp_GenFB_NCC signal (PL system module). The control does not finish a test step correctly. **Error correction** -Check whether a previous system error of the control led to the cancellation of a test section Check the wiring of the signal pp_GenFB_NCC - Inform your service agency

Error number	Description
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error during the test run or in the hardware/wiring. The control does not finish a test step correctly. The control does not perform a certain test. **Error correction** - Check whether a previous system error of the control led to the cancellation of a test section - Check the hardware/wiring (PWM ribbon cable) and exchange it if necessary - Inform your service agency
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error during the test run or in the hardware. The control does not finish a test step correctly. The control does not perform a certain test. **Error correction** - Check whether a previous system error of the control led to the cancellation of a test section - Check the hardware and exchange it if necessary - Inform your service agency
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error during the test run or in the hardware. The control does not finish a test step correctly. The control does not perform a certain test. **Error correction** - Check whether a previous system error of the control led to the cancellation of a test section - Check the hardware/wiring for whether SPL/SMOP A outputs are permanently at a High level - Inform your service agency

Error number	Description
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** The cutout channel test was ended autonomously because of an error during the test run or in the hardware. The control does not finish a test step correctly. The control does not perform a certain test. **Error correction** - Check whether a previous system error of the control led to the cancellation of a test section - Check the hardware/wiring for whether SPL/SMOP B outputs are permanently at a High level - Inform your service agency
237-1F300	**Error message** 1F300 CC%2 Cancelation of the cutout channel test %4 **Cause of error** An error occurred during the test run, or another error occurred; this led to the drive being switched off and cancellation of the brake test. The control does not finish a test step correctly. The control does not perform a certain test. **Error correction** - Check whether a previous system error of the control led to the cancellation of a test section - Check the software version - Inform your service agency
237-3001	**Error message** 13001 UM3: schwerwiegender Fehler (Stopp-Reaktion SS1) %1 **Cause of error** Für weitere Informationen nachfolgende Alarmmeldungen (13xxx) beachten! **Error correction**
237-3002	**Error message** 13002 UM3: Interne Überwachung (Stopp-Reaktion SS2) %1 **Cause of error** Für weitere Informationen nachfolgende Alarmmeldungen (13xxx) beachten! **Error correction**

Error number	Description
237-3007	**Error message** 13007 UM3: VarioLink-Kommunikationsfehler %1 **Cause of error** - Umrichter meldet Fehler in der VarioLink-Kommunikation **Error correction** - Steuerung neu starten - Variolink-Verbindung überprüfen
237-3010	**Error message** 13010 UM3: Umrichter meldet Fehler in der asynchronen Kommunikation %1 **Cause of error** - Umrichter meldet Fehler in der asynchronen Kommunikation mit der CC **Error correction** - Steuerung neu starten - Kundendienst benachrichtigen
237-3011	**Error message** 13011 UM3: CC meldet Fehler in der asynchronen Kommunikation %1 **Cause of error** - CC meldet Fehler in der asynchronen Kommunikation mit dem Umrichter **Error correction** - Steuerung neu starten - Kundendienst benachrichtigen
237-3018	**Error message** 13018 UM3: CC antwortet nicht auf Message %4 vom Umrichter %1 **Cause of error** - CC antwortet innerhalb einer vorgegebenen Zeit nicht auf die angegebene Message vom Umrichter (Message Code ist dezimal angezeigt, aber hexadezimal zu interpretieren) **Error correction** - Steuerung neu starten - Kundendienst benachrichtigen

Error number	Description
237-3019	**Error message** 13019 UM3: Umrichter antwortet nicht auf Kommando %4 von der CC %1 **Cause of error** - Der Umrichter antwortet innerhalb einer vorgegebenen Zeit nicht auf das angegebene Kommando von der CC (Kommando Code ist dezimal angezeigt, aber hexadezimal zu interpretieren) **Error correction** - Steuerung neu starten - Kundendienst benachrichtigen
237-3801	**Error message** 13801 UM-FSSW:Telegrammzähler Fehler FS-Kommunikation %1 Soll:%4 Ist:%5 **Cause of error** **Error correction**
237-3802	**Error message** 13802 UM-FSSW: UM-DriveID Fehler FS-Kommunikation %1 Soll:%4 Ist:%5 **Cause of error** **Error correction**
237-3804	**Error message** 13804 UM-SOC: Topology inspection of the UM parameters failed %1 **Cause of error** **Error correction**
237-3805	**Error message** 13805 UM-SOC: FS configuration data were falsified %1 %4 %5 **Cause of error** **Error correction**
237-3806	**Error message** 13806 UM-SOC: UM-DriveID from UM-SOC and UM-FSuC are unequal %1 %4 %5 **Cause of error** **Error correction**

Error number	Description
237-3820	**Error message** 13820 UM-SOC: Spannungsüberwachung 3,3V-FSuC überschritten **Cause of error** **Error correction**
237-3891	**Error message** 13891 UM-SOC: Fehlerhafte Include-Datei (erhaltene Version: %4 - erwartete Version:%5) **Cause of error** - Software von Reglereinheit und Umrichter(FS) sind nicht mit der gleichen Include-Datei compiliert worden. **Error correction** - Softwareversion prüfen und gegebenenfalls neu laden - Kundendienst benachrichtigen
237-8800	**Error message** 18800 SKERN-CC: Geberwarnung Amplitude zu niedrig X%4 **Cause of error** **Error correction**
238-1000	**Error message** 1000 UVR%2 overcurrent **Cause of error** The power supply unit hat detected an exceedance of the permissible current on its supply connection **Error correction** Inform your service agency
238-1001	**Error message** 1001 UVR%2 phase current too high **Cause of error** The line current consumed by the power supply unit is close to the maximum permissible value **Error correction** - Inspect the design of the inverter system - Reduce the power consumption of the inverter system - Inform your service agency
238-1002	**Error message** 1002 UVR%2 DC-link voltage too high **Cause of error** The power supply unit has detected that the DC-link voltage is too high **Error correction** Inform your service agency

Error number	Description
238-1003	**Error message** 1003 UVR%2 error in power supply **Cause of error** The supply unit hat detected a faulty power supply network. **Error correction** - Check the supply connection, and ensure correct supply connection - Inform your service agency
238-1004	**Error message** 1004 UVR%2 DC-link voltage too low **Cause of error** The power supply unit is reporting that the DC-link voltage is too low. **Error correction** - Check the supply connection - Check the parameters and stability of the supply network - Inform your service agency
238-1006	**Error message** 1006 UVR%2 leakage current too high **Cause of error** The leakage current monitor of the power supply unit has detected an impermissibly high value. **Error correction** - Check the wiring of the power modules and motors - Check the motor lines and DC link for sufficient insulation resistance to ground - Inform your service agency
238-1007	**Error message** 1007 UVR%2 temperature too high at heat sink **Cause of error** The temperature of the heat sink in the UVR exceeds a critical value. **Error correction** Reduce the load
238-1008	**Error message** 1008 UVR%2 error in control of IGBT **Cause of error** The supply unit has detected an error with the IGBT drive circuit **Error correction** Inform your service agency

Error number	Description
238-100A	**Error message** 100A UVR%2 temperature at heat sink critically high **Cause of error** The heat-sink temperature in the UVR supply unit is reaching critical values. **Error correction** - Reduce the power drawn from the DC link - Check the temperature or climate control unit of the electrical cabinet - Check for ventilation clearances around the power supply unit - Inform your service agency
238-100B	**Error message** 100B UVR%2 error in DC-link charge **Cause of error** The power supply unit has detected a fault during charging of the DC link. **Error correction** - Check the DC-link bus mounting or the DC-link wiring for a short circuit - Inform your service agency.
238-100C	**Error message** 100C UVR%2 service mode activated **Cause of error** The service mode of the UVR has been activated. Control of the UVR now occurs over the service interface. **Error correction** End the service mode at the service interface.
238-100D	**Error message** 100D UVR%2 faulty PWM control **Cause of error** The PWM drive circuit monitor is reporting an error **Error correction** Inform your service agency
238-100E	**Error message** 100E UVR%2 hardware detection faulty **Cause of error** The hardware detection of the power supply unit (so-called HIK) is faulty **Error correction** Inform your service agency

Error number	Description
238-100F	**Error message** 100F UVR%2 configuration error **Cause of error** The configuration of the UVR power supply unit is faulty. **Error correction** - Check the configuration of the power supply unit, and correct it as necessary (CfgSupplyModule3xx) - Inform your service agency
238-1010	**Error message** 1010 UVR%2 low voltage faulty **Cause of error** The supply voltage monitor in the supply unit is reporting an error **Error correction** Inform your service agency
238-1011	**Error message** 1011 UVR%2 failure of fan for power supply unit **Cause of error** The UVR power supply unit has detected a fault in the supply unit fan. **Error correction** - Check the supply unit fan for whether it is possibly blocked by objects or contamination - Inform your service agency
238-1012	**Error message** 1012 UVR%2 software error **Cause of error** Internal fault of the power supply unit **Error correction** Inform your service agency
238-1013	**Error message** 1013 UVR%2 error during self-test **Cause of error** A fault has arisen in the power supply unit during the internal self-test. **Error correction** - Check the wiring of the supply unit - Inform your service agency

Error number	Description
238-1014	**Error message** 1014 UVR%2 control error **Cause of error** A fault has arisen in the regulator in the power supply unit **Error correction** Inform your service agency
238-1015	**Error message** 1015 UVR%2 overload in the +24V of the integrated power supply unit **Cause of error** The current consumption of the 24 V consumers on the integrated power supply unit of the UVR exceeds the maximum permissible value. **Error correction** - Check the wiring of the +24 V (X76, X90) of the power supply unit integrated into the UVR - Check consumers on +24 V, and reduce as needed - Check the planning of the machine according to the technical manual of your inverter system - Check the configuration of CfgSupplyModule3xx/MP_ps24VMaxLoadCurr - Inform your service agency
238-1016	**Error message** 1016 UVR%2 CRC error during HSCI transmission **Cause of error** A checksum error was detected during data transmission over HSCI. **Error correction** - Check the HSCI connections and HSCI cables - Inform your service agency
238-1017	**Error message** 1017 UVR %2 maximum permissible DC-link capacity exceeded **Cause of error** The UVR power supply unit has detected an impermissibly high DC-link capacity. **Error correction** - Check the design of the inverter system - Reduce the number of modules connected to the DC-link (UM, CMH) - Inform your service agency

Error number	Description
238-1018	**Error message** 1018 UVR%2 processor temperature reaching critical values **Cause of error** The processor temperature in the UVR is reaching critical values. **Error correction** - Check the temperature in the electrical cabinet - Check the climate control unit for proper functioning - Inform your service agency
238-1019	**Error message** 1019 UVR%2 maximum temperature of processor exceeded **Cause of error** The maximum temperature of the processor in the supply unit has been exceeded. **Error correction** - Check the temperature in the electrical cabinet - Check the climate control unit for proper functioning - Reduce the power of the inverter system - Inform your service agency
238-101A	**Error message** 101A UVR%2 synchronization with line power supply failed **Cause of error** The synchronization between the supply unit and the supply network was not successful. **Error correction** - Check for a faulty connection of the main supply conductors - Check SITOR fuses for the inverter system - Check the power supply network - Inform your service agency
238-101B	**Error message** 101B UVR%2 value of DC-link center voltage too high **Cause of error** The UVR power supply unit has detected that the amount of the DC-link mean voltage is reaching impermissibly high values. **Error correction** - Check the supply voltages of the UVR (line voltage). - Ensure that operation occurs on a TN network. Operation on TT or IT networks is not permitted. Please refer to the technical manual of your inverter system. - Inform your service agency.

Error number	Description
238-101C	**Error message** 101C UVR%2 faulty measurement of voltage **Cause of error** The supply unit is reporting an error in the DC-link voltage measurement **Error correction** Inform your service agency
238-101D	**Error message** 101D UVR%2 configured DC-link voltage too low **Cause of error** The DC-link voltage configured in MP_dcLinkVoltage is too small. The configured DC-link voltage must be greater than the rectifier value of the live line voltage. If a power supply unit is operated, for example, with a line voltage of 3AC 480 V, the machine manufacturer must increase the DC-link voltage to be generated to DC 720 V by means of MP_dcLinkVoltage. **Error correction** - Check the entry in the parameter CfgSupplyModule3xx/MP_dcLinkVoltage and adjust it if necessary. - Inform your service agency.
238-101E	**Error message** 101E UVR%2 temperature sensor is supplying faulty data **Cause of error** The temperature sensor in the supply unit is defective or is returning faulty data. **Error correction** - Inform your service agency
238-101F	**Error message** 101F UVR%2 wrong or no KDR connected **Cause of error** The inductance of the commutating reactor was determined to be too low. Perhaps no commutating reactor is connected, or the connection is faulty. **Error correction** - Check whether the commutating reactor is connected correctly - Check the inductance of the commutating reactor - Inform your service agency

Error number	Description
238-1021	**Error message** 1021 UVR%2 DC-link voltage too low **Cause of error** The power supply unit reports that the DC-link voltage is too low. **Error correction** - Check the power connection - Inform your service agency
238-1022	**Error message** 1022 UVR%2 DC-link voltage low: charging circuit activated **Cause of error** The charging circuit of the power supply unit was activated because the DC-link voltage was too low. **Error correction** - Check the power connection - Inform your service agency
239-0001	**Error message** 1 CC-FSUC: internal error CC%2 %1 2 MC-FSUC: internal error 3 UM-FSUC: internal error CC%2 %1 %10 **Cause of error** Internal software error (unexpected program sequence) **Error correction** - Generate the service file - Inform your service agency
239-0004	**Error message** 4 CC-FSUC: voltage error CC%2 %1 5 MC-FSUC: voltage error 6 UM-FSUC: voltage error CC%2 %1 %10 **Cause of error** Voltage monitoring reports an error. **Error correction** - Check the voltage supply of the affected units ("24 V" status LED on the unit) - Inform your service agency
239-0007	**Error message** 7 CC-FSUC: temperature error CC%2 %1 (temperature: %6, %7°C) 8 MC-FSUC: temperature error (temperature: %6,%7°C) **Cause of error** Temperature monitoring reports an error. **Error correction** - Check the temperature in the electrical cabinet - Inform your service agency

Error number	Description
239-000A	**Error message** A CC-FSUC: parameterization failed CC%2 %1 **Cause of error** The FS microcontroller received invalid parameter data. **Error correction** - Generate the service file - Inform your service agency
239-000C	**Error message** C UM-FSUC: FS parameters inconsistent CC%2 %1 %10 **Cause of error** The FS microcontroller received inconsistent parameter data. **Error correction** - Generate the service file - Inform your service agency
239-000D	**Error message** D CC-FSUC: cyclic communication failed CC%2 %1 **Cause of error** Cyclic communication between the MC and CC FS micro-controller failed. **Error correction** - Generate the service file - Inform your service agency
239-000F	**Error message** F UM-FSUC: cyclic communication failed CC%2 %1 %10 **Cause of error** Cyclic communication between the MC and UM FS micro-controller failed. **Error correction** - Generate the service file - Inform your service agency
239-0010	**Error message** 10 CC-FSUC: software inconsistent CC%2 %1 11 MC-FSUC: software inconsistent 12 UM-FSUC: software inconsistent CC%2 %1 %10 **Cause of error** Internal software error (software corrupt) **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-0013	**Error message** 13 CC-FSUC: stack error CC%2 %1 14 MC-FSUC: stack error 15 UM-FSUC: stack error CC%2 %1 %10 **Cause of error** Internal software error (stack memory) **Error correction** - Generate the service file - Inform your service agency
239-0016	**Error message** 16 CC-FSUC: internal software error CC%2 %1 17 MC-FSUC: internal software error 18 UM-FSUC: internal software error CC%2 %1 %10 **Cause of error** Internal software error (Single Event Upset) **Error correction** - Generate the service file - Inform your service agency
239-0019	**Error message** 19 CC-FSUC: RAM error CC%2 %1 1A MC-FSUC: RAM error 1B UM-FSUC: RAM error CC%2 %1 %10 **Cause of error** Internal software error (RAM memory) **Error correction** - Generate the service file - Inform your service agency
239-001C	**Error message** 1C CC-FSUC: voltage monitoring impaired CC%2 %1 1D MC-FSUC: voltage monitoring impaired 1E UM-FSUC: voltage monitoring impaired CC%2 %1 %10 **Cause of error** The FS microcontroller reports defective monitoring of the voltage. **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-001F	**Error message** 1F CC-FSUC: initial software consistency check CC%2 %1 20 MC-FSUC: initial software consistency check 21 UM-FSUC: initial software consistency check CC%2 %1 %10 **Cause of error** Internal software error (software corrupt) **Error correction** - Generate the service file - Inform your service agency
239-0022	**Error message** 22 CC-FSUC: cyclic communication overloaded CC%2 %1 24 UM-FSUC: cyclic communication overloaded CC%2 %1 %10 **Cause of error** The FS microcontroller has received too many cyclic telegrams. **Error correction** - Generate the service file - Inform your service agency
239-0025	**Error message** 25 CC-FSUC: cyclic communication impaired CC%2 %1 27 UM-FSUC: cyclic communication impaired CC%2 %1 %10 26 MC-FSUC: cyclic communication impaired **Cause of error** Cyclic communication on the FS microcontroller is impaired. **Error correction** - Generate the service file - Inform your service agency
239-002A	**Error message** 2A UM-FSUC: impermissible Reconfiguration CC%2 %1 %10 **Cause of error** The FS microcontroller was reconfigured even though it had already received valid FS configuration data. **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-002D	**Error message** 2D UM-FSUC: device parameters inconsistent CC%2 %1 %10 **Cause of error** The FS microcontroller received inconsistent configuration data. **Error correction** - Generate the service file - Inform your service agency
239-0030	**Error message** 30 UM-FSUC: FS configuration data of B channel invalid CC %2 / %1 10 **Cause of error** Upon request by the B channel, the FS microcontroller set the FS configuration data to invalid. **Error correction** - Generate the service file - Inform your service agency
239-0033	**Error message** 33 UM-FSUC: invalid request for self-test CC%2 %1 %10 **Cause of error** The FS microcontroller received a request to start the self-test even though STO and SBC are not active. **Error correction** - Generate the service file - Inform your service agency
239-0036	**Error message** 36 UM-FSUC: self-test canceled CC%2 %1 %10 **Cause of error** The FS microcontroller canceled a running self-test because of an error. **Error correction** - Generate the service file - Inform your service agency
239-0039	**Error message** 39 UM-FSUC: B channel voltage error CC%2 %1 %10 **Cause of error** The B channel reports a voltage error. **Error correction** - Check the voltage supply of the affected units ("24 V" status LED on the unit) - Inform your service agency

Error number	Description
239-003C	**Error message** 3C UM-FSUC: watchdog B timed out CC%2 %1 %10 **Cause of error** The watchdog of the B channel timed out. **Error correction** - Generate the service file - Inform your service agency
239-003D	**Error message** 3D CC-FSUC: internal watchdog defective CC%2 %1 3E MC-FSUC: internal watchdog defective 3F UM-FSUC: internal watchdog defective CC%2 %1 %10 **Cause of error** Could not configure the internal watchdog of the FS micro-controller. **Error correction** - Generate the service file - Inform your service agency
239-0042	**Error message** 42 UM-FSUC: internal communication error CC%2 %1 %10 **Cause of error** The FS microcontroller detected an error in the cyclic communication within the inverter. **Error correction** - Generate the service file - Inform your service agency
239-0045	**Error message** 45 UM-FSUC: cyclic communication inconsistent CC%2 %1 %10 **Cause of error** The FS microcontroller detected an error in the cyclic communication with the MC. **Error correction** - Generate the service file - Inform your service agency
239-0048	**Error message** 48 UM-FSUC: cyclic communication inconsistent CC%2 %1 %10 **Cause of error** The FS microcontroller detected an error in the cyclic communication with the PAE module. **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-004B	**Error message** 4B UM-FSUC: motor connection wrong CC%2 %1 %10 (E: X %4, P: X%5) **Cause of error** The FS microcontroller detected an incorrect assignment of a motor connection. **Error correction** - Generate the service file - Inform your service agency
239-004E	**Error message** 4E UM-FSUC: implausible parameterization CC%2 %1 %10 **Cause of error** The FS microcontroller detected an incorrect sequence during parameterization. **Error correction** - Generate the service file - Inform your service agency
239-004F	**Error message** 4F CC-FSUC: initialization error CC%2 %1 50 MC-FSUC: initialization error 51 UM-FSUC: initialization error CC%2 %1 %10 **Cause of error** Internal software error (initialization failed) **Error correction** - Generate the service file - Inform your service agency
239-0052	**Error message** 52 CC-FSUC: temperature warning CC%2 %1 (temperature: %6°C) 53 MC-FSUC: temperature warning (temperature: %6°C) **Cause of error** Temperature monitoring reports a warning. **Error correction** - Check the temperature in the electrical cabinet - Inform your service agency
239-0055	**Error message** 55 CC-FSUC: SS1F reaction requested CC%2 %1 57 UM-FSUC: SS1F reaction requested CC%2 %1 %10 **Cause of error** Serious error detected. **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-0058	**Error message** 58 CC-FSUC: incorrect interface version CC%2 %1 59 MC-FSUC: incorrect interface version 5A UM-FSUC: incorrect interface version CC%2 %1 %10 **Cause of error** The interface versions of internal components do not match. **Error correction** - Generate the service file - Inform your service agency
239-005B	**Error message** 5B CC-FSUC: comparison of interface version missing CC%2 %1 5C MC-FSUC: comparison of interface version missing 5D UM-FSUC: comparison of interface version missing CC %2 %1 %10 **Cause of error** The interface versions of internal components must be exchanged. The adjustment has not taken place yet. **Error correction** - Generate the service file - Inform your service agency
239-0101	**Error message** 101 CC-FSUC: internal error CC%2 %1 102 MC-FSUC: internal error 103 UM-FSUC: internal error CC%2 %1 %10 **Cause of error** Internal software error (unexpected program sequence) **Error correction** - Generate the service file - Inform your service agency
239-0111	**Error message** 111 CC-FSUC: voltage error CC%2 %1 112 MC-FSUC: voltage error 113 UM-FSUC: voltage error CC%2 %1 %10 **Cause of error** Voltage monitoring reports an error. **Error correction** - Check the voltage supply of the affected units ("24 V" status LED on the unit) - Inform your service agency

Error number	Description
239-0121	**Error message** 121 CC-FSUC: temperature error CC%2 %1 (temperature: %7, %8°C) 122 MC FSUC: temperature error (temperature: %7,%8°C) **Cause of error** Temperature monitoring reports an error. **Error correction** - Check the temperature in the electrical cabinet - Inform your service agency
239-0131	**Error message** 131 CC-FSUC: parameterization failed CC%2 %1 **Cause of error** The FS microcontroller received invalid parameter data. **Error correction** - Generate the service file - Inform your service agency
239-0133	**Error message** 133 UM-FSUC: FS parameters inconsistent CC%2 %1 %10 **Cause of error** The FS microcontroller received inconsistent parameter data. **Error correction** - Generate the service file - Inform your service agency
239-0141	**Error message** 141 CC-FSUC: cyclic communication failed CC%2 %1 **Cause of error** Cyclic communication between the MC and CC FS micro-controller failed. **Error correction** - Generate the service file - Inform your service agency
239-0143	**Error message** 143 UM-FSUC: cyclic communication failed CC%2 %1 %10 **Cause of error** Cyclic communication between the MC and UM FS micro-controller failed. **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-0151	**Error message** 151 CC-FSUC: software inconsistent CC%2 %1 152 MC-FSUC: software inconsistent 153 UM-FSUC: software inconsistent CC%2 %1 %10 **Cause of error** Internal software error (software corrupt) **Error correction** - Generate the service file - Inform your service agency
239-0161	**Error message** 161 CC-FSUC: stack error CC%2 %1 162 MC-FSUC: stack error 163 UM-FSUC: stack error CC%2 %1 %10 **Cause of error** Internal software error (stack memory) **Error correction** - Generate the service file - Inform your service agency
239-0171	**Error message** 171 CC-FSUC: internal software error CC%2 %1 172 MC-FSUC: internal software error 173 UM-FSUC: internal software error CC%2 %1 %10 **Cause of error** Internal software error (Single Event Upset) **Error correction** - Generate the service file - Inform your service agency
239-0181	**Error message** 181 CC-FSUC: RAM error CC%2 %1 182 MC-FSUC: RAM error 183 UM-FSUC: RAM error CC%2 %1 %10 **Cause of error** Internal software error (RAM memory) **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-0191	**Error message** 191 CC-FSUC: voltage monitoring impaired CC%2 %1 192 MC-FSUC: voltage monitoring impaired 193 UM-FSUC: voltage monitoring impaired CC%2 %1 %10 **Cause of error** The FS microcontroller reports defective monitoring of the voltage. **Error correction** - Generate the service file - Inform your service agency
239-01A1	**Error message** 1A1 CC-FSUC: initial software consistency check CC%2 %1 1A2 MC-FSUC: initial software consistency check 1A3 UM-FSUC: initial software consistency check CC%2 %1 %10 **Cause of error** Internal software error (software corrupt) **Error correction** - Generate the service file - Inform your service agency
239-01B1	**Error message** 1B1 CC-FSUC: cyclic communication overloaded CC%2 %1 1B2 MC FSUC: cyclic communication overloaded 1B3 UM-FSUC: cyclic communication overloaded CC%2 %1 %10 **Cause of error** The FS microcontroller has received too many cyclic telegrams. **Error correction** - Generate the service file - Inform your service agency
239-01C1	**Error message** 1C1 CC-FSUC: cyclic communication impaired CC%2 %1 1C2 MC FSUC: cyclic communication impaired 1C3 UM-FSUC: cyclic communication impaired CC%2 %1 %10 **Cause of error** Cyclic communication on the FS microcontroller is impaired. **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-01D3	**Error message** 1D3 UM-FSUC: impermissible Reconfiguration CC%2 %1 %10 **Cause of error** The FS microcontroller was reconfigured even though it had already received valid FS configuration data. **Error correction** - Generate the service file - Inform your service agency
239-01E3	**Error message** 1E3 UM-FSUC: device parameters inconsistent CC%2 %1 %10 **Cause of error** The FS microcontroller received inconsistent configuration data. **Error correction** - Generate the service file - Inform your service agency
239-01F3	**Error message** 1F3 UM-FSUC: FS configuration data of B channel invalid CC %2 / %1 10 **Cause of error** Upon request by the B channel, the FS microcontroller set the FS configuration data to invalid. **Error correction** - Generate the service file - Inform your service agency
239-0203	**Error message** 203 UM-FSUC: invalid request for self-test CC%2 %1 %10 **Cause of error** The FS microcontroller received a request to start the self-test even though STO and SBC are not active. **Error correction** - Generate the service file - Inform your service agency
239-0213	**Error message** 213 UM-FSUC: self-test canceled CC%2 %1 %10 **Cause of error** The FS microcontroller canceled a running self-test because of an error. **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-0223	**Error message** 223 UM-FSUC: B channel voltage error CC%2 %1 %10 **Cause of error** The B channel reports a voltage error. **Error correction** - Check the voltage supply of the affected units ("24 V" status LED on the unit) - Inform your service agency
239-0233	**Error message** 233 UM-FSUC: watchdog B timed out CC%2 %1 %10 **Cause of error** The watchdog of the B channel timed out. **Error correction** - Generate the service file - Inform your service agency
239-0241	**Error message** 241 CC-FSUC: internal watchdog defective CC%2 %1 242 MC-FSUC: internal watchdog defective 243 UM-FSUC: internal watchdog defective CC%2 %1 %10 **Cause of error** Could not configure the internal watchdog of the FS micro-controller. **Error correction** - Generate the service file - Inform your service agency
239-0253	**Error message** 253 UM-FSUC: internal communication error CC%2 %1 %10 **Cause of error** The FS microcontroller detected an error in the cyclic communication within the inverter. **Error correction** - Generate the service file - Inform your service agency
239-0263	**Error message** 263 UM-FSUC: cyclic communication inconsistent CC%2 %1 %10 **Cause of error** The FS microcontroller detected an error in the cyclic communication with the MC. **Error correction** - Generate the service file - Inform your service agency

Error number	Description
239-0273	**Error message** 273 UM-FSUC: cyclic communication inconsistent CC%2 %1 %10 **Cause of error** The FS microcontroller detected an error in the cyclic communication with the PAE module. **Error correction** - Generate the service file - Inform your service agency
239-0283	**Error message** 283 UM FSUC: motor connection wrong CC%2 %1 %10 (E: X %5, P: X%6) **Cause of error** The FS microcontroller detected an incorrect assignment of a motor connection. **Error correction** - Generate the service file - Inform your service agency
239-0293	**Error message** 293 UM-FSUC: implausible parameterization CC%2 %1 %10 **Cause of error** The FS microcontroller detected an incorrect sequence during parameterization. **Error correction** - Generate the service file - Inform your service agency
239-02A1	**Error message** 2A1 CC-FSUC: initialization error CC%2 %1 2A2 MC-FSUC: initialization error 2A3 UM-FSUC: initialization error CC%2 %1 %10 **Cause of error** Internal software error (initialization failed) **Error correction** - Generate the service file - Inform your service agency
239-02B1	**Error message** 2B1 CC FSUC: temperature warning CC%2 %1 (temperature: %7°C) 2B2 MC FSUC: temperature warning (temperature: %7°C) **Cause of error** Temperature monitoring reports a warning. **Error correction** - Check the temperature in the electrical cabinet - Inform your service agency

Error number	Description
239-02C1	**Error message** 2C1 CC-FSUC: SS1F reaction requested CC%2 %1 2C2 MC FSUC: SS1F reaction requested 2C3 UM-FSUC: SS1F reaction requested CC%2 %1 %10 **Cause of error** Serious error detected. **Error correction** - Generate the service file - Inform your service agency
239-02D1	**Error message** 2D1 CC-FSUC: incorrect interface version CC%2 %1 2D2 MC-FSUC: incorrect interface version 2D3 UM-FSUC: incorrect interface version CC%2 %1 %10 **Cause of error** The interface versions of internal components do not match. **Error correction** - Generate the service file - Inform your service agency
239-02E1	**Error message** 2E1 CC-FSUC: comparison of interface version missing CC %2 %1 2E2 MC-FSUC: comparison of interface version missing 2E3 UM-FSUC: comparison of interface version missing CC %2 %1 %10 **Cause of error** The interface versions of internal components must be exchanged. The adjustment has not taken place yet. **Error correction** - Generate the service file - Inform your service agency
239-FFF1	**Error message** FFF1 CC-FSUC: ALARM TEST CC%2 %1 FFF2 MC-FSUC: ALARM TEST FFF3 UM-FSUC: ALARM TEST CC%2 %1 %10 **Cause of error** The functional safety microcontroller received a test alarm. **Error correction** - Generate the service file - Inform your service agency

Error number	Description
23A-0000	**Error message** Internal error on device: %2 **Cause of error** - Error during program run - Invalid error message %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0001	**Error message** Internal error on device: %2 **Cause of error** - Program or hardware configuration faulty - Test environment is active %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0003	**Error message** Internal access error on device: %2 **Cause of error** HSCI PHY: hardware is not responding %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0004	**Error message** Internal error on device: %2 **Cause of error** - Error during program run (NULL pointer) %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency

Error number	Description
23A-0005	**Error message** Internal error on device: %2 **Cause of error** - Parameter transfer invalid - Illegal value %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0006	**Error message** Internal error on device: %2 **Cause of error** - Parameter transfer invalid - Illegal argument %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0007	**Error message** Error while executing command on device: %2 **Cause of error** Received data invalid %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0008	**Error message** Internal error on device: %2 **Cause of error** Device index invalid %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0009	**Error message** Internal error on device: %2 **Cause of error** Internal registration of function: Error during program run %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency

Error number	Description
23A-000B	**Error message** Timeout during communication with device: %2 **Cause of error** HFL sender: interface at full capacity or not ready for operation %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-000E	**Error message** Timeout during communication with device: %2 **Cause of error** Device is not responding %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0014	**Error message** Faulty data transmission with device: %2 **Cause of error** Received data invalid %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0015	**Error message** Internal error on device: %2 **Cause of error** Interface index invalid %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency

Error number	Description
23A-0018	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL Master: an error occurred at the interface %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0019	**Error message** Error in communication with device:%2 **Cause of error** Inverter (HFL): an error occurred at the interface %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-001A	**Error message** Initialization of interface faulty for device: %2 **Cause of error** SPI: an error occurred at the interface %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-001B	**Error message** Error in communication with device:%2 **Cause of error** SPI: an error occurred at the interface %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-001C	**Error message** Error in communication with device:%2 **Cause of error** SPI: an error occurred at the interface %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-001D	**Error message** Timeout during communication with device: %2 **Cause of error** HSCI (DMA): interface at full capacity or not ready for operation %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-001E	**Error message** Error in communication with device:%2 **Cause of error** HSCI (DMA): an error occurred at the interface %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-001F	**Error message** Error in communication with device:%2 **Cause of error** HFL (DMA): an error occurred at the interface %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0020	**Error message** Timeout during communication with device: %2 **Cause of error** HFL: interface at full capacity or not ready for operation %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0021	**Error message** Internal error on device: %2 **Cause of error** - Error during program run - XADC not initialized %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0022	**Error message** Internal error on device: %2 **Cause of error** - Error during program run - XADC already initialized %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0023	**Error message** Internal error on device: %2 **Cause of error** - Program or hardware configuration faulty - Entry not found in HDT %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency

Error number	Description
23A-0025	**Error message** Initialization of interface faulty for device: %2 **Cause of error** - Device is busy or not ready for operation - Encoder FPGA: configuration failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0026	**Error message** Internal error on device: %2 **Cause of error** Encoder-FPGA Flash ID: program or hardware configuration faulty %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0027	**Error message** Internal error on device: %2 **Cause of error** Encoder FPGA DRAM: program or hardware configuration faulty %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0028	**Error message** Internal access error on device: %2 **Cause of error** Encoder FPGA Flash: hardware is not responding %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0029	**Error message** Initialization of interface faulty for device: %2 **Cause of error** Encoder FPGA Master (HFL): an error occurred at the interface %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-002A	**Error message** Internal error on device: %2 **Cause of error** FSuC bootloader: invalid firmware loaded %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-002B	**Error message** Invalid firmware loaded on device: %2 **Cause of error** FSuC firmware: firmware update failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-002C	**Error message** Invalid firmware loaded on device: %2 **Cause of error** FSuC VMT: firmware update failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-002D	**Error message** Internal error on device: %2 **Cause of error** FSuC HIK: invalid firmware loaded %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-002E	**Error message** Internal error on device: %2 **Cause of error** - Error during program run - FSuC firmware not started %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-002F	**Error message** Error in communication with device:%2 **Cause of error** FSuC: invalid received data %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0030	**Error message** Timeout during communication with device: %2 **Cause of error** FSuC: device is not responding %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0031	**Error message** Timeout during communication with device: %2 **Cause of error** FSuC: device at full capacity or not ready for operation %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0032	**Error message** Firmware update failed on device: %2 **Cause of error** FSuC: invalid firmware loaded %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0033	**Error message** Error in communication with device:%2 **Cause of error** FSuC: invalid parameter transfer %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0034	**Error message** Error in communication with device:%2 **Cause of error** - HSCI: an error occurred at the interface - Local error register is set %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0035	**Error message** Error in communication with device:%2 **Cause of error** - HSCI: an error occurred at the interface - External error register is set %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0036	**Error message** Timeout during communication with device: %2 **Cause of error** HSCI: device is not responding %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0037	**Error message** Error in communication with device:%2 **Cause of error** HSCI: self-test of interface failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0039	**Error message** Internal access error on device: %2 **Cause of error** Flash module: Hardware initialization failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-003A	**Error message** Internal access error on device: %2 **Cause of error** Flash module: access to hardware failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-003B	**Error message** Internal access error on device: %2 **Cause of error** Flash module: access to hardware failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-003C	**Error message** Internal access error on device: %2 **Cause of error** Flash module: access to hardware failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-003D	**Error message** Internal access error on device: %2 **Cause of error** Flash module: access to hardware failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-003E	**Error message** Internal access error on device: %2 **Cause of error** Flash module: invalid parameter transfer %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-003F	**Error message** Internal access error on device: %2 **Cause of error** Flash module: invalid received data %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0040	**Error message** Internal access error on device: %2 **Cause of error** Flash module: configuration of the hardware failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0041	**Error message** Internal access error on device: %2 **Cause of error** Internal access error on device: %2 %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0042	**Error message** Internal error on device: %2 **Cause of error** Zynq initialization: error during program run %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0043	**Error message** Internal error on device: %2 **Cause of error** Start of the application software: error during program run %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0044	**Error message** Internal access error on device: %2 **Cause of error** eFuse status: access to hardware failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0045	**Error message** Internal access error on device: %2 **Cause of error** eFuse key: access to hardware failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0046	**Error message** Internal error on device: %2 **Cause of error** eFuse configuration data: error during program run %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0047	**Error message** Internal access error on device: %2 **Cause of error** eFuse write-operation: access to hardware failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0048	**Error message** Internal access error on device: %2 **Cause of error** RAM: invalid parameter transfer %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0049	**Error message** Internal access error on device: %2 **Cause of error** RAM: hardware test failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-004A	**Error message** Internal access error on device: %2 **Cause of error** FRAM: hardware initialization failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-004F	**Error message** Internal access error on device: %2 **Cause of error** FRAM: hardware is not responding %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0050	**Error message** Internal access error on device: %2 **Cause of error** Diagnostics Flash: hardware initialization failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0052	**Error message** Internal access error on device: %2 **Cause of error** Diagnostics Flash: hardware test failed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0053	**Error message** Internal error on device: %2 **Cause of error** - Error during program run - Invalid image size %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0054	**Error message** Faulty boot image on device: %2 **Cause of error** - Image data are incomplete or inconsistent - Application software not found %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0055	**Error message** Faulty boot image on device: %2 **Cause of error** - Image data are incomplete or inconsistent - Boot image not found %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0056	**Error message** Faulty boot image on device: %2 **Cause of error** - Image data are incomplete or inconsistent - Fall-back boot image not found %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency

Error number	Description
23A-0057	**Error message** Faulty boot image on device: %2 **Cause of error** - Image data are incomplete or inconsistent - Primary boot image not found %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0058	**Error message** Faulty boot image on device: %2 **Cause of error** Fall-back boot image: incorrect image offset %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0059	**Error message** Faulty boot image on device: %2 **Cause of error** Primary boot image: incorrect image offset %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-005A	**Error message** Faulty boot image on device: %2 **Cause of error** Image offset incorrect %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-005B	**Error message** Faulty boot image on device: %2 **Cause of error** Image data are incomplete or inconsistent - Generate the service file and inform your service agency No authentication data are available %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency

Error number	Description
23A-005C	**Error message** Faulty boot image on device: %2 **Cause of error** Byte converter: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-005D	**Error message** Internal error on device: %2 **Cause of error** -Received data invalid - Terminal: invalid character %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-005E	**Error message** Internal error on device: %2 **Cause of error** -Received data invalid - Terminal: input too long %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0060	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0061	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0062	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0063	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0064	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0065	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0066	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0067	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0068	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0069	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-006A	**Error message** Faulty boot image on device: %2 **Cause of error** Info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-006B	**Error message** Internal error on device: %2 **Cause of error** - Device index invalid %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-006C	**Error message** Error while executing command on device: %2 **Cause of error** -Device index invalid - Driver assignment not possible %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-006D	**Error message** Error while executing command on device: %2 **Cause of error** - Device index invalid - Command was not processed %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-006E	**Error message** Internal error on device: %2 **Cause of error** Interface designation invalid %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-006F	**Error message** Internal error on device: %2 **Cause of error** - Error during program run - Error during main initialization phase %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0070	**Error message** Faulty boot image on device: %2 **Cause of error** Image authentication: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0071	**Error message** Error while detecting device for device: %2 **Cause of error** - Program or hardware configuration faulty - Capture limit reached %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0072	**Error message** Error while detecting device for device: %2 **Cause of error** - Image data are incomplete or inconsistent - Faulty entry %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0073	**Error message** Error while detecting device for device: %2 **Cause of error** Device at full capacity or not ready for operation %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0074	**Error message** Internal error on device: %2 **Cause of error** FPGA ID: program or hardware configuration faulty %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency

Error number	Description
23A-0075	**Error message** Error while executing command on device: %2 **Cause of error** - Command invalid - No HIK available for reading out %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0076	**Error message** Internal error on device: %2 **Cause of error** HIK: invalid parameter transfer %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0077	**Error message** Error while executing command on device: %2 **Cause of error** - Parameter transfer invalid - Maximum command size (in) exceeded %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0078	**Error message** Error while executing command on device: %2 **Cause of error** - Parameter transfer invalid - Maximum command size (out) exceeded %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0079	**Error message** Error while executing command on device: %2 **Cause of error** - Parameter transfer invalid - Error in the command sequence %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency

Error number	Description
23A-007A	**Error message** Error while executing command on device: %2 **Cause of error** - Parameter transfer invalid - Address invalid %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-007D	**Error message** Error while executing command on device: %2 **Cause of error** - Parameter transfer invalid - Message ID invalid %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-007E	**Error message** Error while executing command on device: %2 **Cause of error** - Parameter transfer invalid - Command invalid %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-007F	**Error message** Faulty boot image on device: %2 **Cause of error** Encoder FPGA info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency

Error number	Description
23A-0080	**Error message** Faulty boot image on device: %2 **Cause of error** Encoder FPGA info section: image data are incomplete or inconsistent %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0081	**Error message** Internal error on device: %2 **Cause of error** - Program or hardware configuration faulty - Test environment is active %1 ID number: %3, serial number: %4 **Error correction** - Generate the service file and inform your service agency
23A-0082	**Error message** Internal error on device: %2 **Cause of error** - Hardware initialization failed - Internal XADC voltage reference is active %1 ID number: %3, serial number: %4 **Error correction** - Restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0083	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: self-test of interface failed (VMK) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0084	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: self-test of interface failed (VSK) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0085	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: self-test of interface failed (VMLS) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0086	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: self-test of interface failed (VMS) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0087	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: self-test of interface failed (VMPS) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0088	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: self-test of interface failed (VSLPS) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0089	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VMTX) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-008A	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VSTX) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-008B	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VMTO) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-008C	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VSTO) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-008D	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VMRX) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-008E	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VSRX) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-008F	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VMSTX) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0090	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VMI) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0091	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VSI) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0092	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VMSRX) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
23A-0093	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VMPE) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency

Error number	Description
23A-0094	**Error message** Initialization of interface faulty for device: %2 **Cause of error** HFL: an error occurred at the interface (VSPE) %1 ID number: %3, serial number: %4 **Error correction** - Check the interface connections and restart the control - If the problem recurs, generate a service file and inform your service agency
240-07D0	**Error message** No permission to write **Cause of error** Sie haben eine schreibgeschützte Datei zum Editieren angewählt. **Error correction** Vor dem Editieren Schreibschutz aufheben: Schlüsselzahl 86357 eingeben.
240-07D1	**Error message** File '%1' not found **Cause of error** Diese Datei wurde nicht gefunden **Error correction** Datei neu anlegen oder generieren lassen
240-07D2	**Error message** File type of '%1' incorrect **Cause of error** Sie haben eine falsche Datei angewählt **Error correction** Wählen Sie eine andere Datei an
240-07D3	**Error message** File '%1' is encrypted **Cause of error** Sie haben eine verschlüsselte Datei angewählt **Error correction** Geben Sie den Schlüsselcode ein

Error number	Description
240-07D4	**Error message** Access to '%1' is blocked **Cause of error** You tried to edit an NC program while it was running in a Program Run mode. **Error correction** - Stop the NC program run
240-07D5	**Error message** Invalid file path: '%1' **Cause of error** **Error correction**
240-0800	**Error message** Key non-functional **Cause of error** The key is not allowed in this status or it has no function. **Error correction** Press another key or soft key.
240-0804	**Error message** NC program not saved **Cause of error** The NC program is write protected and therefore cannot be saved. **Error correction** - Open the file manager and cancel the write protection for the NC program. Then use the Programming mode of operation to select and save the NC program. - Or, as an alternative, save the NC program under another name.
240-0CA3	**Error message** Error in internal communication **Cause of error** Internal software error. **Error correction** Inform your service agency
240-0CA4	**Error message** Error in an internal process **Cause of error** Internal software error. **Error correction** Inform your service agency

Error number	Description
241-09C4	**Error message** Machine configuration error **Cause of error** A machine parameter has a faulty value. For more information see the soft key INTERNAL INFORMATION (Text0...2). **Error correction** - Correct the machine parameter - Inform your service agency
241-09C5	**Error message** Internal software error **Cause of error** Internal control error: - Insufficient memory - Other, nonspecified errors **Error correction** Inform your service agency.
241-09C6	**Error message** Data record already locked **Cause of error** The table editor was instructed to edit a locked data record. **Error correction** Release the lock (e.g.: terminate the NC program or tool change) and repeat the instruction.
241-09C7	**Error message** Data record with incorrect length **Cause of error** In the table editor, a table was entered containing at least one line whose length differs from the length of the line with the column name. **Error correction** Open the table with the text editor and shorten the problem line at its end or fill it with spaces. Alternatively, for example if several lines are faulty, you can use the file manager to copy the table into a new fault-free table.
241-09C8	**Error message** Pocket table incorrect **Cause of error** - The pocket table contains more spindle pockets than are given in the machine parameter CfgAxes.spindleIndices. **Error correction** - Delete the invalid pockets from the pocket table

Error number	Description
241-09C9	**Error message** Incorrect value \`%1\` in update rule \`%2\` **Cause of error** Incorrect syntax of the update rule: - The keyword is missing or spelled incorrectly - The keyword is unknown - Invalid number for a rule - Incorrect or unknown symbolic table name - The entered column is missing in the table - Column list is unequal during a copying statement **Error correction** - Note the additional information on the error message. - Remember that the error can also be before the indicated place! - Enter the statement in the correct syntax, or contact your machine tool builder.
241-09CA	**Error message** Update rule "%1" erroneous **Cause of error** The update rule for updating a table is faulty and cannot be applied. **Error correction** - Please pay attention to other pending error messages. - Enter the statement for updating the table in the correct syntax, or contact your machine tool builder.
241-09CB	**Error message** Error while importing the table '%1' in '%2' **Cause of error** Table import failed. **Error correction** - Note the additional data on the error message. To do so, press the INTERNAL INFO soft key. - Note further pending error messages. - Inform your service agency.
241-09CC	**Error message** Table '%1' erroneous **Cause of error** The update to the table failed since the table is faulty. **Error correction** Please ensure that the table - has the correct syntax - exists - is not write-protected If necessary, contact your machine tool builder

Error number	Description
241-09CD	**Error message** Error while updating table '%1' **Cause of error** The update of the table failed. **Error correction** - Please pay attention to other pending error messages. - Pay attention to the additional information regarding the error message or contact your machine tool builder.
241-09D0	**Error message** Access to file %1 during update denied **Cause of error** - The file cannot be updated because access to the table is not possible - The file might still be in use by the control **Error correction** - Press NC stop, close all files and try again. - If an update is still not possible: restart the control until the power-interrupted message appears. Then restart the update.
241-09D1	**Error message** File %1 changed through import **Cause of error** The entered file was automatically modified during import: - Program name - Special characters removed - End block inserted **Error correction** - Check the file - Note the changes and check - Use the file only if you feel that it is correct
241-0C03	**Error message** File or file path '%1' invalid **Cause of error** The file name or path for the file operation update is incorrect. Update failed. **Error correction** - Note the additional data on the error message. To do so, press the INTERNAL INFO soft key. - Note further pending error messages. - Check the syntax of the file name and path for the file operation. - Check whether the file and path actually exist. - Inform your service agency.

Error number	Description
241-0C04	**Error message** Error during update in file operation **Cause of error** File operation of an update rule failed. **Error correction** - Note the additional data on the error message. To do so, press the INTERNAL INFO soft key. - Note further pending error messages. - Inform your service agency.
241-0C05	**Error message** Update rule "%1" erroneous **Cause of error** The update rule for the file operation is faulty and has failed. **Error correction** - Note the additional data on the error message. To do so, press the INTERNAL INFO soft key. - Note further pending error messages. - Check the syntax of the command for the file. - Inform your service agency
242-07D1	**Error message** File invalid **Cause of error** - The file to be inserted in the text was not found - No file selected - Selected file is already being edited by the text editor - Selected file is already being edited by another application - Invalid file **Error correction** - Select another file - Close the file in another application
242-07D3	**Error message** Intermediate memory empty **Cause of error** You attempted to insert blocks from intermediate memory, although you haven't copied anything since power has been on. **Error correction** Before you can insert anything from intermediate memory you must first fill it using the copy function.

Error number	Description
242-07D4	**Error message** Text not found Text not found **Cause of error** The ASCII editor could not find the desired text in a file. **Error correction** Search for another text (note upper and lower case letter).
242-07D5	**Error message** Input error **Cause of error** You enter a value that exceeds the permissible input range. **Error correction** Check input value.
242-07D7	**Error message** No permission to write **Cause of error** The called file is write-protected. Sometimes it can no longer be saved under this name. **Error correction** - Save the file under another name - Select another file - Cancel write protection
242-07DA	**Error message** File not saved **Cause of error** The file is write-protected and could not be saved. **Error correction** - Cancel the write-protection in the file management. Select the text editor and save the file again. - Save the file under another name.
243-00F5	**Error message** Serial number? **Cause of error** **Error correction**
245-03F5	**Error message** Configuration server not ready **Cause of error** The system-inherent communication over the data interface to the configuration server is not ready. **Error correction** Inform your service agency.

Error number	Description
245-03F6	**Error message** Unable to open configuration server queue Unable to open configuration server queue **Cause of error** Error in the system-inherent communication. **Error correction** Inform your service agency.
245-03F7	**Error message** Unable to read configuration data '%1' Unable to read configuration data '%1' Unable to write configuration data '%1' **Cause of error** Error in the system-inherent communication. **Error correction** Inform your service agency.
245-03F8	**Error message** Unable to write configuration data '%1' **Cause of error** Error in the system-inherent communication. **Error correction** Inform your service agency.
245-03F9	**Error message** Error in the PGM-MGT configuration %1 **Cause of error** Invalid for incorrect data in the configuration of the file management **Error correction** Correct the corresponding data and save
245-03FA	**Error message** Internal error! **Cause of error** Internal software error in the file management **Error correction** Inform your service agency
245-03FB	**Error message** Process not available **Cause of error** A process entered in the configuration data cannot be activated. **Error correction** Inform your service agency.

Error number	Description
245-03FC	**Error message** ClientQueue (%1) could not be opened **Cause of error** Error in system-inherent communication **Error correction** Inform your service agency.
245-03FD	**Error message** General error in internal system queue (%1) **Cause of error** Error in system-inherent communication **Error correction** Inform your service agency.
245-03FE	**Error message** No receiver for internal system message **Cause of error** Error in system-inherent communication **Error correction** Inform your service agency.
245-03FF	**Error message** Processor check error **Cause of error** Internal software error **Error correction** Inform your service agency
245-0401	**Error message** Failed to send internal message **Cause of error** Error in system-inherent communication **Error correction** Inform your service agency.
245-040D	**Error message** Ext. in-/output not ready **Cause of error** - The interface is not connected - The external device is not switched on or is not ready - The transmission cable is defective or incorrect **Error correction** Check the data transmission line.

Error number	Description
245-0413	**Error message** Problems recognizing the USB device! **Cause of error** - The USB device in use has not been detected - The USB device could not be integrated into the system **Error correction** - Remove the USB device and try again - Try it with another USB device
245-0414	**Error message** Problems removing the USB device! **Cause of error** - The USB device used was not correctly removed or was not logged off by means of the soft-key function - A file is still open on the USB device - The USB device could not be released from the system **Error correction** - Use the soft-key function to log the relevant USB device off correctly - Close the applications with access to a file on the USB device
245-0416	**Error message** Serial data transmission is faulty **Cause of error** - The interface is not correctly configured - The transmission cable is defective or incorrect - Invalid file for the selected interface configuration **Error correction** - Check the interface configuration of both communication partners - Check the data transmission line - Check whether a valid file is available for transmission
245-0851	**Error message** Display program could not be started. **Cause of error** A selection box is still open for starting an external display program **Error correction** Bring the hidden selection box back into the foreground, if necessary with the HeROS task bar - Either select the desired display program for further editing, - or cancel the selection of the display program

Error number	Description
245-0861	**Error message** Changing the access rights is not permitted: %1 **Cause of error** Changing of the access rights to the file or directory is permitted only for the owner or the "root" user. **Error correction**
250-138B	**Error message** Program was edited **Cause of error** The current NC program, or one of the NC programs that have called the current NC program, was changed. It is therefore impossible to go back into the program. **Error correction** - Check whether the changed program should be started Use the mid-program startup function or the GOTO function to select the desired location to go back into the program. HEIDENHAIN recommends using the mid-program startup function to go back into the program.
250-138C	**Error message** Not enough main memory (RAM) **Cause of error** The control has too little physical main memory. **Error correction** Equip the hardware with at least 128 MB RAM.
250-138D	**Error message** Current program not selected. Select with file manager. **Cause of error** The displayed program was not selected in the program run. **Error correction** Use the file manager to select a program at the beginning.
250-138E	**Error message** Program was edited during program run! **Cause of error** A program was edited during program run. **Error correction** Check whether the edited program should be run.

Error number	Description
250-138F	**Error message** Cycle Query dialog box could not be opened! **Cause of error** A dialog box is already open. **Error correction** Close the open dialog box, and restart the program.
250-1390	**Error message** Application cannot be activated: %1 **Cause of error** The other application has not been started or has another title. **Error correction** Start the other application manually.
250-1391	**Error message** Error while transferring command to PLC server Error while transferring command to PLC server **Cause of error** Internal software error. **Error correction** Inform your service agency.
250-1392	**Error message** Error while transferring command to SQL server **Cause of error** Internal software error. **Error correction** Inform your service agency.
250-1393	**Error message** Error in internal communication **Cause of error** Internal software error. **Error correction** Inform your service agency.
250-1394	**Error message** Error in an internal process **Cause of error** Internal software error. **Error correction** Inform your service agency.

Error number	Description
250-1395	**Error message** Error in an internal process **Cause of error** Internal software error. **Error correction** Inform your service agency.
250-1396	**Error message** Program resumption not possible Program resumption not possible **Cause of error** The program cannot be continued from the point of interruption. **Error correction** Use GoTo to position to the start of program, or select the program again. If you are machining a pallet, then update the pallet table (perhaps set W-STATUS to BLANK). Then the program can be started.
250-1397	**Error message** Axis movement canceled **Cause of error** A reference run in an axis was stopped or canceled before the target position was reached. **Error correction** - Check the axis, acknowledge the error, and restart the reference run - If appropriate, press the permissive button to confirm that a non-referenced or non-checked axis is to be moved
250-1398	**Error message** File access not possible **Cause of error** The file cannot be accessed. Perhaps it was deleted. **Error correction** Select another file.
250-1399	**Error message** File access not possible **Cause of error** The file is not a valid NC program for this control. **Error correction** Select another program.

Error number	Description
250-139A	**Error message** NC program not selected **Cause of error** The NC program was not selected through the file management (PGM MGT key). **Error correction** To be able to start this program, select it in the file management (PGM MGT key) of start it with the selected program in mid-program startup.
250-13A7	**Error message** Connection setup to DNC not possible Connection setup to DNC not possible **Cause of error** Cannot connect with the DNC. **Error correction**
250-13A8	**Error message** Connection setup to DNC not possible **Cause of error** Cannot connect with the DNC. TeleService is already being used by another application. **Error correction** - Restart the control. - If the problem continues, inform your service agency.
250-13A9	**Error message** Connection setup to DNC not possible **Cause of error** Cannot connect with the DNC. The machine parameter for TeleService is improperly configured. **Error correction** The machine parameter for TeleService, "CfgServiceRequest," must be correctly configured.
250-13AA	**Error message** TeleService request is in error **Cause of error** TeleService could not be activated or deactivated. **Error correction**

Error number	Description
250-13AB	**Error message** TeleService request is in error TeleService request is in error **Cause of error** Unable to send TeleService request. **Error correction** Check the network connections and retry.
250-13AC	**Error message** TeleService request is in error TeleService request is in error **Cause of error** The machine parameter for TeleService is improperly configured. **Error correction** The machine parameter for TeleService, "CfgServiceRequest," must be correctly configured.
250-13AE	**Error message** Table access not possible **Cause of error** The table cannot be accessed. The file was probably deleted. **Error correction** Make a new table.
250-13AF	**Error message** Preset table is faulty **Cause of error** No preset is marked as active in the preset table, or multiple presets are marked at the same time. **Error correction** Correct the preset table. Only one preset can be marked as active.
250-13B6	**Error message** Configuration parameter cannot be saved **Cause of error** The control tried to write data to a write-protected configuration file. **Error correction** - Inform your service agency

Error number	Description
250-13B7	**Error message** Table cannot be selected **Cause of error** A configuration parameter required for selecting the table could not be written to. **Error correction** - Inform your service agency.
250-13B8	**Error message** Machine kinematic configuration has too many rotary axes **Cause of error** You have defined more than two axes as rotary axes. The working plane cannot be tilted with more that two rotary axes. Machine kinematics with more than two rotary axes are not supported by the control. **Error correction** - Change the machine configuration: Use a kinematic configuration with no more than 2 rotary axes.
250-13BD	**Error message** PLC pop-up window (Module 9216) not possible **Cause of error** A PLC pop-up window (PLC Module 9216) cannot be shown in the current operating situation. **Error correction** Activate the machine operation or close a dialog already open.
250-13BE	**Error message** Tool number does not exist **Cause of error** The tool number required for selection of a pocket number is not in the table. **Error correction** - Correct the tool table
250-13BF	**Error message** Tool does not exist **Cause of error** The tool is not present in the tool table. **Error correction** - Correct the tool table - Select a tool table that contains the tool.

Error number	Description
250-13C0	**Error message** No tool pocket available **Cause of error** There is no fitting tool pocket in the pocket table for the tool. **Error correction** - Provide a fitting tool pocket
250-13C1	**Error message** Pop-up window (Module 9217) not possible **Cause of error** A PLC pop-up window (PLC Module 9217) cannot be shown in the current operating situation. **Error correction** Activate the Machine operating mode or close an already open dialog box.
250-13C2	**Error message** Cycle Query dialog is not possible in this state! **Cause of error** A cycle query dialog (PLC Module 9291) cannot be shown in the current operating situation. **Error correction** - Check the PLC program and correct if necessary
250-13C3	**Error message** Check the "Evaluation of EnDat" dialog window **Cause of error** The "Evaluation of EnDat encoder" dialog window cannot be displayed because is overlapped by a background operating mode. **Error correction** Activate a machine operating mode and confirm the "Evaluation of EnDat encoder" dialog window.
250-13C4	**Error message** Program selection not possible **Cause of error** The selection of an NC program is presently not allowed. **Error correction** Select the Program Run mode of operation

Error number	Description
250-13C6	**Error message** Program start not supported **Cause of error** The start of the NC program is not supported in this condition. **Error correction** - Move the axes over the reference marks - Reconduct the program start
250-13C7	**Error message** Operation not supported **Cause of error** The control cannot be operated while it is in this state. **Error correction** Please wait until the axes have been referenced.
250-13CA	**Error message** Start the program in the Program Run mode **Cause of error** An error occurred in the previous program run **Error correction** Switch to the Program Run mode of operation and restart the program
250-17D3	**Error message** Reference run is not possible **Cause of error** The reference run is not possible at present because the axis is in use. **Error correction** Restart the reference run at a later time.
250-F306	**Error message** Program selection not possible **Cause of error** An external program selection (over DNC, OPC UA, or the PLC) could not be executed because the file manager is open. **Error correction** - Select the program and close the file manager. - Or close the file manager and select the program externally (over DNC, OPC UA, or the PLC).

Error number	Description
250-F308	**Error message** File manager was closed through an external program selection **Cause of error** While the file manager was open, the control received a command for an external program start (over DNC, OPC UA, or the PLC). The file manager was closed and the external program selection was conducted. **Error correction**
250-F30C	**Error message** Preset table is faulty **Cause of error** No preset is marked as active in the preset table, or multiple presets are marked at the same time. **Error correction** Correct the preset table
250-F319	**Error message** Function could not be executed %1 **Cause of error** The function called (e.g. pressing of a soft key) could not be executed. **Error correction** - Call the function again later - If the problem continues, inform your service agency
250-F31A	**Error message** Execution canceled -- next start from beginning of table **Cause of error** - Program aborted by user - Program aborted because of an NC error - Attempt at starting failed because of a missing table entry **Error correction** Remove the cause of the error. The next program start will take place from the beginning of the table.
250-F31D	**Error message** Host computer operation is not possible **Cause of error** Host computer operation cannot be enabled because a machining process is currently running. **Error correction** Terminate the current operation and then activate host computer operation.

Error number	Description
250-F322	**Error message** Illegal program type **Cause of error** The program type of the file is not valid. **Error correction** Select a valid NC program
250-F323	**Error message** 3DROT active: Axis positions cannot be tested **Cause of error** You have attempted to test an axis even though the function "Rotate working plane" is active. Axis positions cannot be tested if the working plane is tilted. **Error correction** Deactivate "Tilt the working plane" and test the axis positions again
250-F324	**Error message** Component %1 cannot be displayed due to limits currently configured **Cause of error** Possible causes: - A component with fewer than four limits was entered in the machine configuration under CfgComponentMon/components[] - The values of the limits are not in ascending order. **Error correction** - Correct the machine configuration under CfgComponentMon/components[] or select another component in the "CM Detail" tab
250-F329	**Error message** Monitoring task %1 cannot be displayed. **Cause of error** Possible causes: - A monitoring task with fewer than four limits was entered in the machine configuration under Monitoring/CfgMonPreferences/monitoringTasks[] - The values of the limits of the entry in CfgMonComponent are not in ascending order. - No value is entered for CfgMonComponent/display. **Error correction** - Check and if necessary correct the machine configuration under Monitoring/CfgMonComponent, or - Select another monitoring task in the "MON Detail" tab

Error number	Description
250-F32A	**Error message** Table cannot be activated **Cause of error** Activation of a different datum or compensation table is not possible while a block is being machined. **Error correction** Either wait until the active block has finished, or (if possible without danger) press the "Internal stop" soft key while the block is being machined.
250-F32E	**Error message** Monitoring task %1 cannot be displayed. **Cause of error** The monitoring task cannot be displayed. Possible causes: - The display settings for this monitoring task are incomplete or faulty - The configured limits do not permit display of the monitoring task **Error correction** - Check the configuration and adapt it if necessary: CfgMonComponent > display - Select a different monitoring task
250-F332	**Error message** Program cannot be run **Cause of error** Either there is an error in the program, or the program has been modified. **Error correction** - Check whether the program is free of errors, and execute a GOTO or reset before restarting - Use the GOTO function in order to select the desired starting point for the restart. Or use a reset to restart the simulation from the beginning of the program.
250-F333	**Error message** Preset was not stored **Cause of error** An error occurred while saving the preset. **Error correction** - Check the preset table and correct it if necessary - Restart the control - Try to save the preset again - If the error recurs, save the service file and inform your service agency

Error number	Description
250-F334	**Error message** Preset was not reset **Cause of error** An error occurred while resetting the preset. **Error correction** - Check the preset table and correct it if necessary - Restart the control - Try to save the preset again - If the error recurs, save the service file and inform your service agency
250-F335	**Error message** This file cannot be displayed **Cause of error** Access to the file was denied. **Error correction** Check the access rights for the file
250-F33C	**Error message** Check the "Evaluation of EnDat" dialog window **Cause of error** The "Evaluation of EnDat encoder" dialog window can't be opened, because the "Traverse reference points" dialog box is not active. **Error correction** Activate the "Traverse reference points" dialog box and confirm the "Evaluation of EnDat encoder" dialog window.
250-F33D	**Error message** GPS: settings have not been saved yet **Cause of error** You edited entries in the global program settings but did not save these entries. **Error correction** Apply or discard the settings
251-0D92	**Error message** Communication error when reading the configuration data Unexpected answer when reading the active parameter set Login for reading the configuration data has failed **Cause of error** An internal communication error has occurred in the internal oscilloscope during interrogation of the configuration data. **Error correction** - Check the configuration of the parameter set and correct if necessary - Inform your service agency

Error number	Description
251-0D95	**Error message** Internal communication error in oscilloscope **Cause of error** An internal communication error has occurred in the internal oscilloscope during log-in to the Channel Manager. **Error correction** Inform your service agency.
251-0D96	**Error message** Internal communication error in oscilloscope **Cause of error** An internal communication error has occurred in the internal oscilloscope during registration for operating modes. **Error correction** Inform your service agency.
251-0D97	**Error message** Step function has been deactivated **Cause of error** Due to an operating mode change, the jump function was deactivated in the integrated oscilloscope. **Error correction** Run the jump function in manual mode
251-0D98	**Error message** Incorrect operating mode was selected for step function **Cause of error** The jump function can be activated only in manual mode. **Error correction** Change to the manual operating mode
251-0D99	**Error message** Parameter set cannot be read **Cause of error** Interrogation of the active parameter set of an axis has failed. **Error correction** Check the parameter sets of the axis
251-0D9B	**Error message** Initializing parameters cannot be read **Cause of error** An error occurred during import of the initialization parameters for the jump function of the active axis. **Error correction** Check the initialization file

Error number	Description
251-0D9C	**Error message** Error while restoring the parameters **Cause of error** The restoration of and the original parameters of an axis has failed. **Error correction** Check the axis parameters after deactivation of the jump function
251-0D9D	**Error message** Parameter-set code of an axis does not exist **Cause of error** The identifier of the parameter set for restoring the original axis parameters is missing. **Error correction** After deactivation of the jump function, check the parameters of the axis and correct them if necessary
251-0D9F	**Error message** Step function cannot be activated **Cause of error** The reference marks for one or more axes were not scanned. **Error correction** - Home the axes - Reselect the jump function in the integrated oscilloscope
251-0DA0	**Error message** Step function cannot be activated **Cause of error** An internal communication error has occurred. The status of the axis's parameter set or the status of the reference run could not be ascertained. **Error correction** - Reselect the jump function in the integrated oscilloscope - Inform your service agency if the error occurs repeatedly
251-0DA1	**Error message** Parameter set cannot be initialized **Cause of error** The active parameter set of an axis could not be initialized for the jump function. **Error correction** - Reselect the jump function in the integrated oscilloscope - Inform your service agency if the error occurs repeatedly

Error number	Description
251-0DA6	**Error message** Step function cannot be activated **Cause of error** An error occurred during import of the initialization parameters or the parameter sets for the jump function. **Error correction** Check the parameter sets and initialization parameters, and correct if required.
251-0DA7	**Error message** Actual-to-nominal value transfer is faulty **Cause of error** No actual-to-nominal value transfer could be executed after deactivation of the jump function. **Error correction** Inform your service agency if this error occurs repeatedly
251-0DB7	**Error message** The selected signal is not supported **Cause of error** The selected position signal is not supported by the control hardware. **Error correction** Choose another signal.
251-0DD5	**Error message** Selected CC signal is not available **Cause of error** When the data record is started, the selected signal can not be assigned to the CC. **Error correction** If the error occurs repeatedly, please inform your service agency.
251-0DD6	**Error message** Error in synchronization of data channels to the PLC **Cause of error** Synchronization error between IPO and PLC at start of data recording. Signals from the PLC are to be recorded, but the IPO does not receive a response from the PLC. **Error correction** If the error occurs repeatedly, please inform your service agency.

Error number	Description
251-0DD7	**Error message** Error in synchronization of CC signals **Cause of error** Synchronization error between IPO and CC at start of data recording. CC signals are to be recorded, but the IPO does not receive a response from the CC. **Error correction** If the error occurs repeatedly, please inform your service agency.
251-0DD8	**Error message** Number of CC signals exceeded **Cause of error** The permissible number of CC signals has been exceeded. **Error correction** Limit the number of CC signals to the permissible maximum.
251-0DD9	**Error message** Unexpected sampling rate during acknowledgment of CC data channel **Cause of error** When a CC data channel is acknowledged, the cycle time of the acknowledgment does not correspond to the expected cycle time. **Error correction** Check the parameter settings for the cycle time.
251-0DDA	**Error message** Number of data channels for recording with CC clock exceeded. **Cause of error** When recording with the CC clock, the number of data channels is limited. **Error correction** Limit the number of data channels when recording with CC clock to the permissible maximum.
251-0DE2	**Error message** The selected signal is not supported **Cause of error** The right to IPO-/CC-Dbg signals is missing **Error correction**

Error number	Description
251-0DE3	**Error message** The signal cannot be recorded **Cause of error** The right needed to record IPO or CC-Dbg signals is missing. **Error correction** Log on to the control as a user who has the right to read OEM interface data (e.g. as the function user 'oemdataaccessread').
251-0DE4	**Error message** The signal cannot be recorded **Cause of error** The right to record PLC signals is missing **Error correction**
251-0DE5	**Error message** The application cannot be started **Cause of error** The right to run the internal control oscilloscope is missing **Error correction**
251-0DE6	**Error message** The selected signal is not supported **Cause of error** The right to PLC signals is missing **Error correction**
251-0E18	**Error message** Check data of the SCO file **Cause of error** You opened an older version of a SCO file; this file is not compatible with the current version. **Error correction** Check the loaded data.
251-0E28	**Error message** The trace file could not be saved **Cause of error** Trace file cannot be written to. **Error correction** Check whether write-access is granted for the selected partition (e.g. PLC).

Error number	Description
251-0E29	**Error message** File could not be read **Cause of error** A file could not be accessed for reading. **Error correction** Check whether read-access is granted for the selected partition (e.g. PLC).
251-0E2A	**Error message** Necessary right for recording a signal is missing **Cause of error** When loading a file or initializing the control, changed user rights have caused a signal to not have the necessary rights for recording The signal is set to OFF in the selection dialog box of the integrated oscilloscope. **Error correction**
251-0E2B	**Error message** Signal ID not found in list of enabled signals **Cause of error** The signal ID used was not found in the list of permitted signals. The signal is set to OFF in the selection dialog box of the integrated oscilloscope. **Error correction**
251-0E2C	**Error message** Recording of signal not permitted **Cause of error** There is no recording right for this signal. The signal is set to OFF in the selection dialog box of the integrated oscilloscope. **Error correction**
251-0E2D	**Error message** Signal ID not found in selection list **Cause of error** The signal ID used was not found in the list of of the selection dialog box of the integrated oscilloscope. The signal is set to OFF. The registered user might not have the right to access the signal. **Error correction** Log-on as a user with the necessary access right

Error number	Description
251-0E34	**Error message** Selected UVR signal is not available **Cause of error** When data recording is started, the selected signal of the UVR cannot be assigned. **Error correction** If the error occurs repeatedly, please inform your service agency.
251-0E35	**Error message** Error in synchronization of CC or UVR signals **Cause of error** Synchronization error between interpolator, CC controller unit, or UVR at start of data recording. CC or UVR signals are to be recorded, but the interpolator receives no response from the CC or UVR. **Error correction** If the error occurs repeatedly, please inform your service agency.
251-0E36	**Error message** Trigger condition Signal [SAVED] not fulfilled in reference trace **Cause of error** A signal was set to [SAVED] even though a trigger condition has not been fulfilled for the reference trace. **Error correction** Before the signal state [SAVED] can be used, a trigger condition must be fulfilled for the reference trace, otherwise the signal curve cannot be associated chronologically.
251-0E37	**Error message** Trigger condition Signal [SAVED] not fulfilled in current trace **Cause of error** The signal state [SAVED] is being used even though no trigger condition has been fulfilled for the current trace. **Error correction** If the signal state [SAVED] is to be used, then a trigger condition must be fulfilled for the current trace. Otherwise the signal curve cannot be associated chronologically.
260-01FB	**Error message** Cannot find PLC module: %1 **Cause of error** Cannot find the selected PLC module. **Error correction** Select another PLC module or inform your service agency.

Error number	Description
260-01FC	**Error message** Error in temporary file **Cause of error** The temporary file could not be generated. **Error correction** Select another PLC program or inform your service agency.
260-01FD	**Error message** Compiler not found **Cause of error** The PLC compiler / PET interpreter could not be loaded. **Error correction** Inform your service agency.
260-01FE	**Error message** Program/table with invalid format **Cause of error** The PLC program / the PET table has an incorrect format. **Error correction** Correct the program/table.
260-01FF	**Error message** PLC compiler configuration incorrect **Cause of error** The configuration file for the PLC compiler contains errors. **Error correction** Correct the configuration file. Note further error messages regarding this.
260-0200	**Error message** Insufficient memory for the PLC compiler/PET **Cause of error** Insufficient memory to load the PLC compiler / PET. **Error correction** Release some memory and compile again.

Error number	Description
260-0202	**Error message** System error in the PLC **Cause of error** The following causes of error are intended to help the service personnel to isolate and correct the PLC system error. - A PLC program cannot be loaded because the control type is configured incorrectly. - A PLC program cannot be loaded because the error table for the PLC compiler is configured incorrectly. - A PLC program cannot be loaded because an unexpected file system error occurred. - Symbol definitions for the PLC program cannot be read or contain unexpected data. - There is another internal PLC system error **Error correction** Inform your service agency.
260-0203	**Error message** Command cannot be executed at present **Cause of error** The command cannot be executed at present because either an NC program is running or a compiler process is already active. **Error correction** Stop the program run and restart the compiler process.
260-0204	**Error message** The command cannot be executed at present **Cause of error** The command cannot be executed at present because autostart is already active. **Error correction** End autostart and restart the compiler process.
260-0205	**Error message** PLC program could not be started **Cause of error** The PLC program was compiled correctly but the PLC could not be started. **Error correction** Select another PLC program and compile it; or inform your service agency.

Error number	Description
260-0206	**Error message** No active PLC program **Cause of error** The PLC program has not started and therefore no program for tracing is active. **Error correction** Select another PLC program and compile it; or inform your service agency.
260-0207	**Error message** No access to PLC symbol file (%1) **Cause of error** A PLC symbol file cannot be accessed. **Error correction** Compile the PLC program again or select another PLC program.
260-0208	**Error message** Local symbol. Cannot be displayed **Cause of error** **Error correction**
260-0209	**Error message** No valid PLC operand **Cause of error** The input contains no valid PLC operands. **Error correction** Please enter the correct PLC operand name.
260-020A	**Error message** Update of operand data not possible **Cause of error** The operand data update is not possible at present. **Error correction** Compile the PLC-Programm again or inform your service agency.
260-020B	**Error message** Source file invalid **Cause of error** **Error correction**

Error number	Description
260-020C	**Error message** No active error table **Cause of error** The error table is not active. **Error correction** Reselect the error table and compile.
260-020D	**Error message** Compilation of PLC program failed **Cause of error** The PLC program could not be compiled. **Error correction** Select another PLC program and compile.
260-020E	**Error message** Compilation of PLC error table failed **Cause of error** The error table could not be compiled. **Error correction** Select another error table and compile.
260-020F	**Error message** Operand being loaded in watchlist.... **Cause of error** **Error correction**
260-0210	**Error message** Operand not recognized! **Cause of error** **Error correction**
260-0211	**Error message** Operand %1 loaded in watchlist **Cause of error** **Error correction**
260-021D	**Error message** Command not possible at this time: Trace function is active **Cause of error** The command cannot be executed at present. An external tracer is active, which is why the command is not permissible. **Error correction** End the external tracer and give the command again.

Error number	Description
260-021E	**Error message** Command not possible at this time: Self-test is active **Cause of error** The command cannot be executed at present. The safety self-test is active, which is why the command is not permissible. **Error correction** Wait for the end of the self-test, and then give the command again.
260-0221	**Error message** Operand %1 not activated/deactivated **Cause of error** The given operand could not be activated or deactivated in the I/O force list. **Error correction** Please check the I/O force list and correct it if necessary.
260-0224	**Error message** Operand %1 was set twice **Cause of error** The given operand was entered more than once in the I/O force list **Error correction** - Adapt selection. PLC operands of the same name should be listed only once in the I/O force list. - If there are multiple entries, the last operand in the list applies!
260-0235	**Error message** Data request not permitted **Cause of error** Due to missing user rights, no data can be requested from the PLC. **Error correction** - Check the user rights - If necessary, grant the current user the necessary additional rights
270-0001	**Error message** System error in SQL server **Cause of error** A software error has occurred in the SQL server. **Error correction** Inform your service agency.

Error number	Description
270-0002	**Error message** System error in SQL server **Cause of error** A function of the SQL server has been called, although the function has not yet been implemented. **Error correction** Inform your service agency.
270-0003	**Error message** System error in SQL server **Cause of error** The SQL server cannot determine the sender of a message. **Error correction** Inform your service agency.
270-0004	**Error message** System error in SQL server **Cause of error** The SQL server cannot contact the sender of a message. **Error correction** Inform your service agency.
270-0005	**Error message** Symbolic name %1 is not resolvable **Cause of error** A given symbolic table name cannot be resolved with the configuration data. **Error correction** – Check the configuration data - Contact your machine tool builder
270-0006	**Error message** Table %1 not found **Cause of error** A table file cannot be found in the given path. **Error correction** - Correct the given path name - Copy or move the file to the given location

Error number	Description
270-0007	**Error message** Error while accessing file %1 **Cause of error** An error occurred while accessing the indicated file: - The file is not ISO Latin-15 encoded and begins with a Byte Order Mark (BOM) - The file contains impermissible control characters, particularly the string 'NUL' - Reading of the file is not permissible - The file is to be deleted, but may not be written - The file is to be renamed, but may not be written - A previous access to the already open file failed - There are already too many open files in the file system - The file system does not permit the creation of any more files - The file system reports some other error **Error correction** - Check the encoding of the file, change it if necessary, and remove the Byte Order Mark - Check whether the file contains impermissible control characters and remove them - Check and adapt the access rights to this file if necessary - Check the status of the file system, and repair the file system if necessary
270-0008	**Error message** Table %1 erroneous **Cause of error** A given table file contains incorrect syntax in the table description. **Error correction** Edit the table description so that it has, in this sequence and spelling, the keyword BEGIN, the table name, and optionally the keyword MM or INCH for the unit of measure; or contact your machine tool builder.
270-0009	**Error message** Table %1 incomplete **Cause of error** A give table file does not end with the keyword [END]. This file might be incomplete. **Error correction** - Check whether the table file is complete. If necessary, add the keyword [END] on its own line at the end of the file - If necessary, contact your machine tool builder

Error number	Description
270-000A	**Error message** Definition of column %1 is faulty or missing **Cause of error** The table contains a column for which either there is no description in the configuration data or in the table itself, or whose description in the table is faulty. **Error correction** - Add to or correct the column description - If necessary, contact your machine tool builder
270-000B	**Error message** Field name %1 already assigned **Cause of error** A given table file contains one field name more than once. **Error correction** - Correct the table - If necessary, contact your machine tool builder
270-000C	**Error message** Syntactically incorrect SQL statement **Cause of error** A given SQL statement has incorrect syntax. **Error correction** - Enter the statement in the correct syntax - If necessary, contact your machine tool builder
270-000D	**Error message** Literal not concluded **Cause of error** A literal in a given SQL statement does not end with the required ' character. **Error correction** Enter the statement in the correct syntax, or contact your machine tool builder.
270-000E	**Error message** Field name %1 not found **Cause of error** An SQL statement was given with a field name that is not contained in the table. **Error correction** - Enter the statement in the correct syntax - If necessary, contact your machine tool builder

Error number	Description
270-000F	**Error message** Data record already locked **Cause of error** There was an attempt to lock access to or to edit a data record that was already locked from somewhere else. **Error correction** - Remove the external lock and repeat the statement - If necessary, contact your machine tool builder
270-0010	**Error message** Data record with incorrect length **Cause of error** A given table contains at least one data record whose length differs from the length of the line with the field names. **Error correction** - Shorten the data record or fill it with space characters - If necessary, contact your machine tool builder
270-0011	**Error message** No further data records found **Cause of error** No further data records were found in response to an SQL query. **Error correction** Reformulate the query if the desired data record has not already been found.
270-0012	**Error message** Incorrect default value for field **Cause of error** There was an attempt to insert a data record in a table, where an incorrect default value was given for at least one field in the description. **Error correction** - Correct the default value. In most cases it cannot be saved in the available field length. - If necessary, contact your machine tool builder
270-0013	**Error message** Incorrect value for field **Cause of error** There was an attempt to edit a data record in a table, where an incorrect value was given for at least one field. **Error correction** - Correct the given value. Usually it cannot be saved in the available field length. - If necessary, contact your machine tool builder

Error number	Description
270-0014	**Error message** Incorrect number of values **Cause of error** There was an attempt to insert or change a data record in a table, where the number of values do not match the number of selected fields. **Error correction** - Correct the statement - If necessary, contact your machine tool builder
270-0015	**Error message** System error in SQL server **Cause of error** Unknown error in the SQL server. **Error correction** Inform your service agency.
270-0016	**Error message** Table synonym already exists **Cause of error** An attempt was made to generate for the table a logical name that already exists. **Error correction** Select another name or delete the existing name first.
270-0017	**Error message** Table synonym not found **Cause of error** There was an attempt to delete or edit a nonexistent logical name for a table. **Error correction** Select another name.
270-0018	**Error message** Table already exists **Cause of error** An attempt was made to create a new table, although a table with this name already exists. **Error correction** Select another table name.

Error number	Description
270-0019	**Error message** Table is still open **Cause of error** An attempt was made the change a table description or delete a table, although the table is still open. **Error correction** Close the table first.
270-001A	**Error message** Configuration datum cannot be modified **Cause of error** An attempt was made to create or change a logical name for a table. However, the corresponding configuration datum cannot be written. **Error correction** Stop the machining and then try again to generate or change the logical name.
270-001B	**Error message** Description of the columns cannot be read **Cause of error** No description of the columns can be found for the table, because the type of table is not included in the configuration data, or the description of columns saved in the table itself is incomplete or syntactically incorrect. **Error correction** Check the configuration data and add a description of the type of table. Open the table with a text editor and delete or edit the column description saved in the table.
270-001C	**Error message** Table contains no columns **Cause of error** An attempt was made to make or open a table without columns. **Error correction** Delete the table and make a new one.
270-001D	**Error message** Table is write-protected **Cause of error** There was an attempt to make or edit a table on a write-protected storage medium or on one marked as write protected. **Error correction** Remove the write protection.

Error number	Description
270-001E	**Error message** Column %1 already exists in table **Cause of error** An attempt was made in a table to insert an existing column twice. **Error correction** Enter another column name.
270-001F	**Error message** Index already defined **Cause of error** An attempt was made to create an index for a table using an index name that was already defined. **Error correction** Check the SQL statement and enter a different index name.
270-0020	**Error message** Index not found **Cause of error** An attempt was made to delete an index that does not exist. **Error correction** Enter a valid name.
270-0021	**Error message** Value %1 is not unique **Cause of error** The indicated value appears more than once in a column configured for unique values. **Error correction** Change the values in the column such that each value is unique.
270-0022	**Error message** Column %1 cannot be indexed **Cause of error** An attempt was made to create an index for a column that was not configured for unique values. **Error correction** Change the values in the column such that each value is unique and configure the column for unique values, or enter a different column for indexing.

Error number	Description
270-0023	**Error message** Caption of the column %1 too long **Cause of error** The key name given for the column configuration is just as long as or longer than the given column width. **Error correction** - Enter a larger value for the column width - If necessary, contact your machine tool builder
270-0024	**Error message** Configuration datum %1 - %2 contains no value **Cause of error** The designated attribute of the column configuration contains an empty string. **Error correction** - Enter a valid value - Delete the attribute if no default value is needed - If necessary, inform your machine manufacturer
270-0025	**Error message** Configuration datum %1 - %2 contains syntactically incorrect value %3 **Cause of error** The designated attribute of the column configuration contains a syntactically incorrect value. **Error correction** - Enter a valid value - If necessary, inform your machine manufacturer
270-0026	**Error message** Configuration datum %1 - %2 contains a value %3 that is too long **Cause of error** The designated attribute of the column configuration contains a value that cannot be entered in the column with the given width **Error correction** - Enter a larger value for the column width - Enter another value for the attribute - If necessary, inform your machine manufacturer

Error number	Description
270-0027	**Error message** Unnecessary configuration datum %1 - %2 **Cause of error** The indicated attribute for the column configuration is not defined for the data type of the column. **Error correction** Delete the attribute or contact your machine tool builder.
270-0028	**Error message** Configuration datum %1 - %2 contains value %3 out of range **Cause of error** The designated attribute of the column configuration lies outside of the value range. It might, for example, be smaller than the minimum value. **Error correction** - Enter a valid value - If necessary, inform your machine manufacturer
270-0029	**Error message** Configuration of table type %1 refers to undefined column %2 **Cause of error** The column designated in the configuration of the table type is not defined. **Error correction** - Check the column name - Delete the column - Delete the column from the configuration of the table type - If necessary, contact your machine tool builder
270-002A	**Error message** Primary key of table type %1 refers to undefined column %2 **Cause of error** The designated primary key is not a column of the table type. **Error correction** - Check the primary key - Enter the column designated in the list of columns as the primary key of the table - If necessary, inform your machine manufacturer

Error number	Description
270-002B	**Error message** Foreign key of table type %1 refers to undefined column %2 **Cause of error** The indicated foreign key is not a column of this table type. **Error correction** Check the entry for the foreign key, or enter a column identified in the list of columns as the foreign key for this table, or contact your machine tool builder.
270-002C	**Error message** Foreign key of the table type %1 contains invalid referential action %3 **Cause of error** Incorrect syntax in the referential action for the designated foreign key. **Error correction** - Check the given referential action - If necessary, inform your machine manufacturer
270-002D	**Error message** Configuration datum %1 - %2 contains errors **Cause of error** The indicated configuration datum contains errors. The erroneous values were replaced by default values for operating the SQL server. **Error correction** - Correct the indicated configuration datum - Check whether the primaryKey parameter has the correct initial value for the table column - If necessary, inform your machine manufacturer
270-002E	**Error message** Configuration datum %1 - %2 contains errors **Cause of error** The given configuration datum has errors and was not accepted for operating the SQL server. **Error correction** - Correct the given configuration datum - If necessary, inform your machine manufacturer

Error number	Description
270-002F	**Error message** No column given **Cause of error** The SQL does not specify a column. **Error correction** Enter at least one column in the statement.
270-0030	**Error message** Primary key may not be altered **Cause of error** The column for the primary key may not be renamed or removed from the table. **Error correction** Check the entered SQL statement.
270-0031	**Error message** Update %1 not possible **Cause of error** An attempt was made to change a value in the column indicated as primary key for the table. **Error correction** Check the entered SQL statement.
270-0032	**Error message** Value of primary key %1 missing **Cause of error** An attempt was made to enter a line in the table, even though there was no value entered in the primary key column of the table. **Error correction** Check the entered SQL statement.
270-0033	**Error message** Column %1 contains no value **Cause of error** An attempt was made to delete a value from the indicated column. However, a value must be entered in this column. **Error correction** Check the entered SQL statement and assign a valid value to the column.

Error number	Description
270-0034	**Error message** Erroneous value for %1 **Cause of error** An attempt was made to enter a syntactically incorrect value to the column. **Error correction** Check the entered SQL statement and assign a valid value to the column.
270-0035	**Error message** Value for %1 is too long **Cause of error** An attempt was made to enter a value wider than the column width to the column. **Error correction** Check the entered SQL statement and assign a valid value to the column.
270-0036	**Error message** Erroneous value for %1 **Cause of error** An attempt was made to enter a value outside the value range to the column. **Error correction** Check the entered SQL statement and assign a valid value to the column.
270-0037	**Error message** Data record %1 cannot be inserted, changed or deleted **Cause of error** There was an attempt to insert, change or delete a data record that either makes reference through a foreign key to a non-available data record or to which at least one other data record makes reference through a foreign key. The specified referential action would leave an invalid reference. **Error correction** Check the SQL statement and correct the references first.
270-0038	**Error message** Data record %1 cannot be changed **Cause of error** There was an attempt to change a data record that uses a foreign key to refer to another data record. The specified referential action would change a data record that was already changed by the SQL statement itself. **Error correction** Check the SQL statement and correct the references first.

Error number	Description
270-0039	**Error message** Data record already deleted **Cause of error** There was an attempt to access a data record that was already deleted by another statement. **Error correction** - Correct the statement - If necessary, contact your machine tool builder
270-003A	**Error message** Error in table name or table type %1 **Cause of error** The given table name or type has incorrect syntax. Table names and types must begin with a letter and can otherwise contain letters, numerals and the underscore character. **Error correction** Check and correct the table name or type.
270-003B	**Error message** Data medium almost full **Cause of error** When table files were closed, it was impossible to save a compact version of the files due to insufficient space remaining on the data medium. **Error correction** Delete any unnecessary files on the data medium.
270-003C	**Error message** The modification key does not designate a time stamp column **Cause of error** A modification key was entered in the table configuration, but the corresponding column does not have the type TSTAMP. **Error correction** Select the column type TSTAMP or configure another time stamp column as modification key.
270-003D	**Error message** The modification key designates a read only column **Cause of error** A modification key was entered in the table configuration and the corresponding column was declared only readable. This is not allowed because this column must always take the current time stamp for changes. **Error correction** Remove READONLY in the column configuration or configure another column as modification key.

Error number	Description
270-003E	**Error message** The modification key designates a unique column **Cause of error** A modification key was entered in the table configuration and the corresponding column was declared unique. This is not allowed because the same time stamp might be saved in two or more lines when changes are close together in time. **Error correction** Remove UNIQUE in the column configuration or configure another column as modification key.
270-003F	**Error message** Merging of incompatible tables **Cause of error** Two tables were entered for merging that have different basic types. The basic types are derived from the file footers and headers in the tables. **Error correction** Check the file footers and headers of both table files and their configuration. If the tables have different basic types, they cannot be merged.
270-0040	**Error message** Column %1 for special functions is not allowed **Cause of error** The column given for a special function (primary key, foreign key, timestamp or password) does not exist, has the wrong file type or is too narrow. Further information: - Columns for a foreign key have to have the same data type as the column for the primary key. - The column for the timestamp must have the TSTAMP data type and be at least 19 characters wide. - The column for the password must have the TEXT data type and be at least 15 characters wide. - The columns for the timestamp and password must not be write-protected or ambiguous. **Error correction** Correct the description of the table type or the column in the configuration data

Error number	Description
270-0041	**Error message** Data record %1 is protected by password **Cause of error** You tried to delete, edit or cancel the password protection of a table's password-protected data record. **Error correction** - To change the data record, reenter the password in the password column or cancel the password protection. - Cancel the password protection before deleting the data record. - To cancel the password protection, enter the exclamation mark character "!" in the password column and immediately enter the password.
270-0042	**Error message** Data record %1 was changed without authorization **Cause of error** The checksum calculated for table's data record does not agree with the stored checksum. The table file was manipulated by an external application or damaged on the storage medium. **Error correction** Import a backup copy of the affected table file.
270-0043	**Error message** The table file %1 might have been manipulated **Cause of error** Not all data records in the table file have checksums saved in the column for the password. Data records might have been changed by an external application. **Error correction** Check all data records for correctness or import a backup copy of the table file.
280-0064	**Error message** FN 14: error code %1 **Cause of error** Error forced by the function FN14 (ISO: D14). This function calls preprogrammed messages of the machine tool builder (for example from an OEM cycle). If with FN14 (D14) is reached during a program run or test run, the program is interrupted and the corresponding message is displayed. **Error correction** Look for a description of the error in the machine tool manual. Restart the program after the cause of the error has been removed.

Error number	Description
280-03E9	**Error message** Tool axis is missing **Cause of error** You programmed a positioning block with tool radius compensation without first calling a tool. **Error correction** Edit the part program.
280-03EE	**Error message** Rotation not permitted **Cause of error** Rotation was programmed before a touch probe cycle. **Error correction** Reset the rotation cycle.
280-03EF	**Error message** Scaling factor not permitted **Cause of error** A scaling factor was programmed before a touch probe cycle. **Error correction** Reset the scaling factor or the axis-specific scaling cycles.
280-03F0	**Error message** Mirroring not permitted **Cause of error** Mirroring was programmed before a touch probe cycle. **Error correction** Reset the mirroring cycle.
280-03F1	**Error message** Datum shift not permitted **Cause of error** Datum shift is active. **Error correction** Reset the datum shift.
280-03F4	**Error message** Wrong sign programmed **Cause of error** The programmed dwell time in the Dwell Time cycle, Peck Drilling cycle, or Tapping cycle is negative (through Q parameter). **Error correction** Edit the cycle parameter.

Error number	Description
280-03F6	**Error message** Touch point inaccessible **Cause of error** No touch point was reached during the TCH-PROBE 0 cycle (ISO: G55) or when using the manual probing cycles. **Error correction** - Pre-position the touch probe closer to the workpiece.
280-03F8	**Error message** Contradictory entry Contradictory entry **Cause of error** The values that you entered are contradictory. **Error correction** Check the input values.
280-03FC	**Error message** Wrong rpm Wrong rpm **Cause of error** The programmed spindle speed does not lie in the existing pattern of spindle speed stages. **Error correction** Enter the correct rotational speed.
280-03FD	**Error message** Radius comp. undefined **Cause of error** You programmed a radius-compensated single-axis positioning block which without the radius compensation does not result in tool movement (e.g. IX+0 R+, ISO: G7). **Error correction** Edit the part program.
280-03FE	**Error message** Rounding-off not permitted **Cause of error** In the positioning block before a rounding arc (RND, ISO: G25) either you programmed a movement only in the tool axis or you used the M function M98 to cancel compensation. **Error correction** Edit the part program.

Error number	Description
280-0400	**Error message** Program start undefined **Cause of error** Type of interpolation undefined. **Error correction** Restart the part program.
280-0401	**Error message** Excessive subprogramming **Cause of error** You nested more than 8 subprogram calls (CALL LBL xx, ISO: Lx,0). **Error correction** Check whether all your subprograms are concluded with LBL 0 (ISO:G98 L0).
280-0413	**Error message** No datum table active **Cause of error** Probing cycle for datum setting: The measured point is to be written in a datum table. However, you have not activated a datum table in a program run mode. **Error correction** In the Program Run, Single Block or Program Run, Full Sequence mode, activate a datum table in which the point is to be written.
280-042C	**Error message** Datum table? **Cause of error** A datum table is required to run an NC program. However, there is no table saved in the control's NC memory, or there are more than one table but none of them is activated. **Error correction** Activate a datum table in the Program Run, Full Sequence operating mode.
280-0430	**Error message** Tolerance exceeded Tolerance exceeded **Cause of error** The limits entered in the tool table TOOL.T in the LTOL or RTOL column were exceeded. **Error correction** Check the limit values for the active calibration tool.

Error number	Description
280-0431	**Error message** Mid-program startup active Mid-program startup active **Cause of error** Mid-program startup is not permitted with the programmed function. **Error correction** Mark the programmed function with "skip blocks" and activate this setting. Then run the mid-program startup again.
280-0432	**Error message** ORIENTATION not permitted **Cause of error** - Your machine does not offer spindle orientation - Spindle orientation not possible **Error correction** - Refer to your machine manual! - Check machine parameter mStrobeOrient and enter the numerical designation of the M function or -1 for spindle orientation by the NC. Refer to your machine manual!
280-0433	**Error message** 3-D ROT not permitted 3DROT not permitted **Cause of error** You tried to conduct one of the following functions while the working plane was tilted: - Setting a reference point - A touch probe cycle 40x for measuring a misalignment **Error correction** Deactivate the tilted working plane function and restart the program.
280-0434	**Error message** Activate 3DROT Activate 3DROT **Cause of error** In the Manual operating mode, the Tilt Working Plane function is inactive. **Error correction** Activate 3DROT in Manual mode.

Error number	Description
280-0435	**Error message** Check the depth sign Check the depth sign **Cause of error** The cycle can only run in negative direction (positive in Cycle 204) because the configuration datum "displayDepthErr" is set to on. **Error correction** - Enter the depth as a negative value (positive in Cycle 204) to run the cycle - To run the cycle in the positive direction (negative in Cycle 204,), set the configuration datum "displayDepthErr" to off
280-0436	**Error message** Q303 in meas. cycle undefined! Q303 in meas. cycle undefined! **Cause of error** In one of the measuring cycles 410 to 418, you did not define the parameter Q303 (measured value transfer)(current value = -1). For reasons of security, however, selection of the measured value transfer is required to write the results of measurement in a table (datum table or preset table). **Error correction** Change parameter Q303 (measured value transfer) in the measuring cycle that is causing the error: - Q303=0: Write the measured values with respect to the active workpiece coordinate system into the active datum table (activate in the program with Cycle 7!). - Q303=1: Write the measured values with respect to the active machine-based coordinate system (REF values) into the preset table (activate in the program with Cycle 247!). - Q303=-1: Measured value transfer is undefined. This value is automatically generated by the TNC when you download a program that was written on a TNC 4xx or with an old software level of the iTNC 530, or if during the cycle definition you skipped the prompt for the measured value transfer with the END key.
280-0437	**Error message** Tool axis not allowed Tool axis not allowed **Cause of error** - You called probing cycle 419 with an illegal tool axis. - You called the PATTERN DEF function in connection with an illegal tool axis. **Error correction** - Only call probing cycle 419 with the tool axis X, Y or Z. - Use the PATTERN DEF function only with the tool axis Z (TOOL CALL Z).

Error number	Description
280-0438	**Error message** Calculated values incorrect Calculated values incorrect **Cause of error** In probing cycle 418, the TNC calculated too large a value. You may have defined in an incorrect sequence for the four holes to be probed. **Error correction** Check the probing sequence. Refer to the User's Manual for Touch Probe Cycles.
280-0439	**Error message** Contradictory measuring points Contradictory measuring points **Cause of error** - In one of the probing cycles 400, 403 or 420, you defined a contradictory combination of measuring points and measuring axes. - The selection of measuring points in Cycle 430 results in division by 0. **Error correction** - For measuring axis = reference axis (Q272=1), the parameters Q264 and Q266 are defined at different values. - For measuring axis = minor axis (Q272=2), the parameters Q263 and Q265 are defined at different values. - For measuring axis = probe axis (Q272=3), the parameters Q263 and Q265 or Q264 and Q266 are defined at different values. - Select the measuring points so that they always have different coordinates in all axes.
280-043A	**Error message** Incorrect clearance height! Incorrect clearance height! **Cause of error** In Cycle 20 (ISO: G120), you entered a clearance height (Q7) that is lower than the coordinate of the workpiece surface (Q5). **Error correction** Enter a clearance height (Q7) that is higher than the coordinate of the workpiece surface (Q5).

Error number	Description
280-043B	**Error message** Contradictory plunge type! Contradictory plunge type! **Cause of error** The plunging strategy defined in Cycles 251 to 254 contradicts the defined plunging angle of the active tool. **Error correction** Change parameter Q366 in one of the Cycles 251 to 254 or the plunging angle ANGLE of the tool in the tool table. Permissible combinations of parameter Q366 and the plunging ANGLE are: For perpendicular plunging: Q366 = 0 and ANGLE = 90 For helical plunging: Q366 = 1 and ANGLE > 0 For inactive tool table, define Q366 with 0 (only perpendicular plunging allowed).
280-043C	**Error message** This fixed cycle not allowed This fixed cycle not allowed **Cause of error** - You attempted to run a fixed cycle in connection with Cycle 220 or 221 although it cannot be combined with these cycles. - You tried to run the Cycle 209 with a feed rate factor for retraction (Q403). **Error correction** - You cannot combine Cycles 220 and 221 with the following fixed cycles: - Cycles of the SLI and SLII groups - Cycles 210 and 211 - Cycles 230 and 231 - Cycle 254 - Feed rate factor 403 is allowed only with a setting in MP3010 unequal to 7.

Error number	Description
280-043D	**Error message** Line is write-protected Line is write-protected **Cause of error** - You tried to edit or erase a write-protected line in the preset table. - You tried to write a value in the active line of the preset table. **Error correction** - Overwriting the active preset is not allowed. Use another preset number. - The write protection was activated by your machine manufacturer. Maybe fixed datum was defined in this line. If you want to cancel write protection, contact your machine tool builder. - You defined the write protection in the TNC.SYS file. If required, cancel the write-protection there. - You tried to change line 0. It cannot be changed.
280-043E	**Error message** Oversize greater than depth Oversize greater than depth **Cause of error** SL cycles II or milling cycles 25x: You have entered an allowance for floor greater than the milling depth. **Error correction** - SL cycles II: Check Q4 in Cycle 20 (ISO: G120). - Milling cycles 25x: Check allowance Q369 and depth Q201.
280-043F	**Error message** No point angle defined Point angle not defined **Cause of error** In Cycle 240 Centering you defined parameter Q343 such that centering is done with respect to the diameter. In a drilling cycle you defined parameter Q395 such that the depth is in reference to the tool diameter. You programmed a cycle for chamfering. The point angle for this must be between 1 and 179 degrees. However, no point angle is defined for the active tool. **Error correction** - Set parameter Q343=0 (centering to entered depth). - Set parameter Q395=0 (depth in reference to the tool tip). - Define the point angle in the column T-ANGLE of the tool table TOOL.T.

Error number	Description
280-0440	**Error message** Contradictory data Contradictory data **Cause of error** The combination of the parameters Depth (Q201) and Diameter (Q344) defined in Cycle 240 Centering under Select depth/diameter (Q343) is not allowed. **Error correction** Possible definitions: Q343=1 (entered diameter active): Q201 must be equal to 0 and Q344 must not be equal to 0. Q343=0 (entered depth active): Q201 must not be equal to 0 and Q344 must be equal to 0.
280-0441	**Error message** Slot position 0 not allowed! Slot position 0 not allowed! **Cause of error** You tried to run Cycle 254 with the slot position 0 (Q367=0) in conjunction with the point pattern Cycle 221. **Error correction** Use slot position Q367 = 1, 2 or 3 if you want to run Cycle 254 with the point pattern cycle 221
280-0442	**Error message** Enter an infeed not equal to 0 Enter an infeed not equal to 0. **Cause of error** You defined a fixed cycle with the depth 0. **Error correction** Enter a depth unequal to 0.
280-0443	**Error message** Switchover of Q399 not allowed **Cause of error** You tried to switch on the touch probe cycle 441 to switch on the angle tracking, although this function is deactivated by configuration datum. **Error correction** In the probe table, set the angle tracking in the TRAC column to ON and then recalibrate the touch probe.

Error number	Description
280-0447	**Error message** Software option not active Software option not active **Cause of error** You tried to use a software option that is not enabled on your TNC. **Error correction** Contact your machine tool builder or the control manufacturer to purchase the software option.
280-0448	**Error message** Kinematics cannot be restored Kinematics cannot be restored **Cause of error** You tried to restore kinematics that do not match the currently active kinematics. **Error correction** Restore only kinematics that you have saved previously from an identical kinematics description.
280-0449	**Error message** Function not permitted Function not permitted **Cause of error** You tried to use a feature that is not enabled on your TNC by the Feature Content Level (FCL) management. **Error correction** By default, FCL functions are locked after a software update. By entering the code number 65535 in the SIK menu, you can enable these functions for a certain period of time for test purposes. You can enable FCL functions permanently by purchasing and entering a code number. For more information, contact your machine tool builder or the control manufacturer.
280-044A	**Error message** Contradictory workpc. blank dim. Contradictory workpc. blank dim. **Cause of error** The workpiece blank dimensions you have defined in a fixed cycle are smaller than the dimensions of the finished part. **Error correction** Check the cycle definition and correct the input values.

Error number	Description
280-044B	**Error message** Measuring position not allowed Measuring position not allowed **Cause of error** The kinematic measurement resulted in a measuring position of 0° in one of the three rotary axes. This is not allowed. **Error correction** Select the starting angle, stopping angle and, if applicable, number of measurements on all three axes in a way that does not result in any 0° positions.
280-044C	**Error message** Kinematic access not possible **Cause of error** The control could not access the active kinematic description (read or write). - There is no valid kinematic description. - The kinematic description is write-protected. **Error correction** - Use a valid kinematic description. - Cancel write-protection for a kinematic description.
280-044D	**Error message** Meas. pos. not in traverse range **Cause of error** You have defined a measured position that lies outside of the rotary axis's traverse range. **Error correction** Select the start angle and/or end angle in the cycle so that the measured position lies within the traverse range.
280-044E	**Error message** Preset compensation not possible **Cause of error** You tried to run a compensation of the preset although not all the entries required for it are in the kinematic description. You can only run the preset compensation if transformations in three separate axes in the machine coordinate system are entered in the kinematic description. **Error correction** Change the number of measuring points in the cycle so that the control can run a preset compensation. If necessary, contact the machine manufacturer.

Error number	Description
280-044F	**Error message** Tool radius too large **Cause of error** - The dimensions you defined in one of the Cycles 251 to 254 are too small. - The lateral oversize in one of the Cycles 251 to 254 is too large. **Error correction** - Use a smaller tool - Reduce the oversize.
280-0450	**Error message** Plunging type is not possible **Cause of error** In one of the cycles 251 to 254 you defined a plunging strategy that is not possible with the dimensions defined in the cycle. **Error correction** Use a smaller tool or another plunging strategy. Set the configuration datum suppressPlungeErr to "on" in order to suppress this monitoring, or use RCUTS in the tool table. Use a tool with a sufficiently large cutting width and define this width in the RCUTS column of the tool table.
280-0451	**Error message** Plunge angle incorrectly defined **Cause of error** You defined an incorrect plunge angle (ANGLE column in the tool table) for the selected plunging strategy. **Error correction** Define a plunge angle greater than 0° and smaller than 90°.
280-0452	**Error message** Angular length is undefined **Cause of error** You defined in the cycle an angular length of 0°. **Error correction** Define an angular length greater than 0° in the cycle definition.

Error number	Description
280-0453	**Error message** Slot width is too large **Cause of error** You defined the circular slot width at least as large as the pitch circle diameter. **Error correction** Enter a slot width smaller than the pitch circle diameter in the cycle definition.
280-0454	**Error message** Scaling factors not equal **Cause of error** You attempted to scale a circular contour element with differing axis-specific scaling factors. **Error correction** Scale the axes of circular contour elements with the
280-0455	**Error message** Tool data inconsistent **Cause of error** A tool whose data do not match those calibrated by the touch probe is active. **Error correction** Transmit the calibrated data of the touch probe to the tool table and perform a TOOL CALL in order to load the changed data.
280-0456	**Error message** MOVE not possible **Cause of error** In a KinematicsOpt cycle you selected a rotary axis positioning with the MOVE function although that is not possible with the existing configuration. **Error correction** Deactivate the MOVE function: - Enter a retraction height Q408 greater than 0. - Define the retraction height Q408 high enough to allow rotary movements without collisions.

Error number	Description
280-0457	**Error message** Presetting not allowed! **Cause of error** - You tried to save a datum in the preset table, although this function is locked by machine parameter. - Touch Probe Cycle 403: Setting the datum in the axis you select is not allowed. It would lead to errors during the tilting of the working plane or in connection with TCPM (M128). **Error correction** - Set the machine parameter MP7295 = 0 for the axes X, Y and Z. If necessary, consult with your machine tool builder. - Check the compensation axis you selected. A datum can be set in the compensation axis only in the first rotary table axis (seen from the workpiece).
280-0458	**Error message** Thread angle too small! **Cause of error** Sum of overrun and pitch is larger than the thread length. **Error correction** - Increase the thread length (in Cycle 831 the length of the overrun is as large as the pitch)
280-0459	**Error message** 3-D ROT status is contradictory! **Cause of error** The 3-D Rot status for the MANUAL and AUTOMATIC operating modes does not match. **Error correction** In 3-D Rot in the MANUAL and AUTOMATIC operating modes, set the same status (ACTIVE/INACTIVE).
280-045A	**Error message** Configuration is incomplete **Cause of error** The configuration data are not prepared for this application. **Error correction** - Inform your machine tool builder.
280-045B	**Error message** No turning tool is active **Cause of error** A turning tool is required for the executed function. **Error correction** - Insert a turning tool (defined via TYPE column in the tool table).

Error number	Description
280-045C	**Error message** Tool orientation is inconsistent **Cause of error** The tool orientation TO does not agree with the selected operation. **Error correction** Check the entry for the tool orientation of the turning tool and the selection (AXIAL / RADIAL) of the cycle being used.
280-045D	**Error message** Angle not possible! **Cause of error** The entered angle is not possible. A taper angle cannot be 0 or 180 degrees. **Error correction** - Correct the value for the entered angle. - Enter a taper angle between 0 and 180 degree, or between 0 and -180 degrees.
280-045E	**Error message** Radius too small! **Cause of error** The radius of the programmed circle is too small. Thread milling: The offset for countersinking on the face is too small. **Error correction** - Check the programmed values for the circle block. - Thread milling: Program an offset greater than 0 for countersinking on the face.
280-045F	**Error message** Thread runout too short! **Cause of error** The thread runout is too short. The minimum length is calculated as: thread runout * thread depth / safety clearance. **Error correction** - Increase the value for the thread runout.

Error number	Description
280-0460	**Error message** Contradictory meas. points **Cause of error** The measured points result in two parallel lines: cannot calculate an intersection. Cannot calculate a straight line from identical measuring points. **Error correction** Choose measuring points so that all the individual points have different coordinates. Two measuring points on a straight line have to have different coordinates.
280-0461	**Error message** Too many limits **Cause of error** Too many limits were selected for face milling. **Error correction** Set no more than 3 limits for face milling.
280-0462	**Error message** Machining strategy with limits not possible **Cause of error** Spiral machining strategy: face milling with limits is not possible **Error correction** Deactivate the limits or select another machining strategy.
280-0463	**Error message** Machining direction not possible **Cause of error** The machining direction is not possible in compliance with the overlap factor. **Error correction** Select another machining direction.
280-0464	**Error message** Check the thread pitch! **Cause of error** The programmed thread pitch differs from the thread pitch of the active tool. **Error correction** Check the value of the thread pitch (PITCH column) in the tool table. If the value of the thread pitch for the active tool is 0, there is no monitoring.

Error number	Description
280-0465	**Error message** Angle cannot be calculated **Cause of error** The inclination angle cannot be calculated. There is no appropriate tilting axis configured in the kinematic description. The programmed inclination angle lies outside of the tilting axis's traverse range. The indexable insert of the tool has been rotated. **Error correction** Check the programmed angle and the preferred direction. Ensure that no rotation (SPB-INSERT) is entered in the tool table.
280-0466	**Error message** Eccentric turning not possible **Cause of error** Control is not configured for eccentric turning. A coupling function is already active through the machine manufacturer. **Error correction** The machine manual provides further information. Check the entry in the configuration datum eccLimSpeedFactor.
280-0467	**Error message** No milling tool is active **Cause of error** The active tool is not defined as a milling cutter. **Error correction** In the tool table, check the entry in the TYPE column. Cycle 880: The hob must be defined as a milling cutter. Cycle 292: Without option 50, a turning tool must also be defined as a milling cutter.
280-0468	**Error message** Insufficient length of cutting edge **Cause of error** Die angegebene Schneidenlänge des aktiven Werkzeugs ist für die Bearbeitung nicht ausreichend. **Error correction** Definieren Sie in der Spalte LCUTS der Werkzeugtabelle die Länge der Werkzeugschneiden. Prüfen Sie bei Zyklus 880 den Eintag in Q553. Ist LCUTS gleich 0 so überwacht Zyklus 880 keine Schneidenlänge.

Error number	Description
280-0469	**Error message** Gear definition is inconsistent or incomplete **Cause of error** Module, tooth number and head diameter are incomplete or contradictory **Error correction** You have to enter at least 2 of the 3 parameters for module/ tooth number/tip circle diameter (not equal to 0). Check the module, tooth number and tip circle diameter , because the given values are contradictory.
280-046A	**Error message** No finishing allowance provided **Cause of error** Es ist kein Aufmaß programmiert, obwohl im Bearbeitungsumfang nur Schlichten angewählt ist. **Error correction** Definieren Sie für die Schlichtbearbeitung ein Aufmaß.
280-046B	**Error message** Line does not exist in table **Cause of error** The programmed action cannot be executed because the given table line does not exist. **Error correction** Check your NC program. Create the given line in your preset or datum table.
280-046C	**Error message** Probing process not possible **Cause of error** There is no workpiece touch probe in the spindle. No direction or path is defined for probing. **Error correction** Insert a touch probe in the spindle. Select the desired angle probing direction by soft key.
280-046D	**Error message** Coupling function not possible **Cause of error** The commanded coupling function cannot be performed. **Error correction** A coupling function has already been activated by the machine tool builder. The kinematic design of the axis to be coupled is not supported. The machine manual provides further information.

Error number	Description
280-046E	**Error message** Fixed cycle is not supported by this NC software **Cause of error** The programmed machining cycle is not supported by this control. **Error correction** Cycle 290 Interpolation turning: - Adapt the NC program - Use Cycle 291 or 292
280-046F	**Error message** Touch probe cycle is not supported by this NC software **Cause of error** The programmed touch probe cycle is not supported by this NC software. **Error correction** Replacement for Cycle 441: Fast probing: Assign a line from the touch-probe table with the desired properties to a tool index.
280-0470	**Error message** NC program aborted **Cause of error** The NC program was aborted through operator action. **Error correction** If required, check the data in your NC program. After the cancellation of the NC program, continue with NC start.
280-0471	**Error message** Touch probe data incomplete **Cause of error** The data of the touch probe are incomplete or incorrectly defined. **Error correction** Check the entries of the touch probe table(TYPE column).
280-0472	**Error message** LAC function not possible **Cause of error** The LAC function is not configured for this axis. **Error correction** Check whether the axis concerned exists (CfgChannelAxis-->progAxis). Check whether LAC is activated for the axis concerned (CfgControllerComp-->enhancedComp). Contact your machine tool builder.

Error number	Description
280-0473	**Error message** Rounding radius or chamfer is too large! **Cause of error** Input parameter Q220: Rounding radius or chamfer is too large **Error correction** Check parameter Q220 and correct the input value if required.
280-0474	**Error message** Axis angle not equal to tilt angle **Cause of error** Probing function not permitted while working plane is inactive: the position of the tilted axes is not equal to 0°. Probing function not permitted while working plane is active: the position of the tilted axes does not match the active angular values. **Error correction** Working plane is inactive: move the tilting axes to the home position. Working plane is active: move the tilting axes to the correct position or adapt the angular values.
280-0475	**Error message** Character height not defined **Cause of error** The transfer value in the input parameters Q513 “Character height” and Q574 “Text length” is 0. **Error correction** In the input parameter Q513, define the desired character height. In the input parameter Q574, define the maximum permissible text length. Define the value 0 for character height in Q513 if you want to scale the engraving to the value defined for text length in Q574.
280-0476	**Error message** Excessive character height **Cause of error** The programmed engraving with the defined character height Q513 results in a longer text than is defined in Q574. **Error correction** Reduce the character height in Q513 so that the programmed engraving is no longer than defined in Q574. Set the value for Q574 higher or to zero in order to make a longer engraving possible. Enter the value 0 in Q513 if you want to scale the engraving to the text length set in Q574.

Error number	Description
280-0477	**Error message** Tolerance error: Workpiece rework **Cause of error** The probed dimension on the workpiece is outside of the defined tolerance. Too little material was removed. The workpiece can be reworked. **Error correction**
280-0478	**Error message** Tolerance error: Workpiece scrap **Cause of error** The probed dimension on the workpiece is outside of the defined tolerance. Too much material was removed. The workpiece is scrap. **Error correction**
280-0479	**Error message** Faulty dimension definition **Cause of error** The definition of a measure or a tolerance cannot be interpreted. **Error correction** Note the rules for defining a measured or tolerance.
280-047A	**Error message** Illegal entry in compensation table **Cause of error** There is a faulty entry in the corresponding compensation table. - The AXIS columns must not refer to linear axes. - The PLC columns must contain no entries. **Error correction** Adapt the configuration or the contents of the compensation table.

Error number	Description
280-047B	**Error message** Transformation not possible **Cause of error** Some transformations between the working plane and basic coordinate system are not allowed for the function performed. Rotations between the tool cutting edge and tool spindle, for example in the tool carrier, are not allowed. **Error correction** Remove the basic rotation and mirroring between the working plane and basic coordinate system. Try out rotations between the tool cutting edge and the tool spindle.
280-047C	**Error message** Tool spindle incorrectly configured **Cause of error** There is an error in the tool spindle configuration. **Error correction** Contact your machine tool builder. Check whether the spindle is listed in CfgAx-es/spindleIndices. Check the attributes “progKind” and “dir” in CfgProgAxis.
280-047D	**Error message** Offset of the turning spindle unknown **Cause of error** A offset set in the rotary spindle for the milling operation cannot be considered for the necessary coupling of the eccentric turning. An offset can be defined by the workpiece datum, pallet datum or PLC datum. **Error correction** If you can ensure that no offset is needed, you can continue machining. In order to consider the offset, the configuration must be changed. Contact your machine tool builder. The rotary spindle must be included as axis in turning mode in the programmable axes (CfgChannelAxes/progAxes or CfgKinSimpleModel/progAxes). This rotary axis must make reference to the active channel spindle (CfgProgAxis/relatedAxis).

Error number	Description
280-047E	**Error message** Global program settings are active **Cause of error** The selected function is not possible with active global program settings. **Error correction** Deactivate the global program settings in order to perform the selected function.
280-047F	**Error message** Faulty configuration of OEM macros **Cause of error** Only one of the two macros for interpolation turning was configured. **Error correction** Contact your machine tool builder. Configure the macro either under CfgSystemCycle OEM_INTERPTURN_ON and OEM_INTERPTURN_OFF, or don't use either of the two macros.
280-0480	**Error message** The combination of programmed oversizes is not possible **Cause of error** A combination of programmed oversizes is not possible. **Error correction** Define either an equidistant oversize or a longitudinal and transverse.
280-0481	**Error message** Measured value not captured **Cause of error** No measured value was ascertained within a probing function. **Error correction** Check whether the probing process was performed. The points can only be evaluation only if a probing process has been successfully conducted.
280-0482	**Error message** Check the monitoring of the tolerance **Cause of error** A consideration of the tolerance is not possible due to an inconsistent working plane. **Error correction** Check the measurement result or correct the settings for the tilted working plane.

Error number	Description
280-0483	**Error message** Hole is smaller than the stylus tip **Cause of error** The diameter of the stylus tip is greater than the diameter of the hole to be measured. **Error correction** Use a smaller stylus tip to measure this hole.
280-0484	**Error message** Preset cannot be set **Cause of error** A correct reference point cannot be written due to an inconsistent machining plane. **Error correction** Correct the setting for tilting the working plane. This monitoring is active due to the configuration of the machine parameter CfgPresetSettings.chkTiltingAxes. If necessary, contact your machine tool builder.
280-0485	**Error message** Alignment of a rotary table is not possible **Cause of error** There is no suitable rotary table in the active machine kinematic model. The axis of the rotary table is not perpendicular in the current workpiece coordinate system. **Error correction** Make sure that there is a rotary table axis with which you can align the workpiece. If necessary, check whether a 3-D basic rotation prevents meaningful alignment.
280-0486	**Error message** Alignment of rotary axes is not possible **Cause of error** Aligning rotary axes to a determined plane is supported only if the basic rotation is also adopted. Aligning a rotary table is not supported if the determined angle is also to be used as a basic rotation. **Error correction** Check the values in the input parameters Q1121 and Q1126.

Error number	Description
280-0487	**Error message** Infeed limited to length of cutting edge **Cause of error** If a cutting edge length is specified in the LCUTS column in TOOL.T, the TNC restricts the infeed to this value. **Error correction** Check the value of the cutting edge length (LCUTS in TOOL.T) and the programmed infeed. Enter the value 0 as the cutting length to switch off this monitoring.
280-0488	**Error message** Machining depth defined as 0 **Cause of error** There is no machining because the machining depth has been programmed with the value zero. **Error correction** Program the machining depth with a value unequal to zero.
280-0489	**Error message** Tool type is unsuitable **Cause of error** In the tool table, a tool type that is not suitable for this operation is defined in the TYP column. **Error correction** Check and correct the entry in the tool table
280-048A	**Error message** Finishing allowance not defined **Cause of error** No machining operation will be performed, since neither an allowance for the side nor an allowance for the depth was programmed for the finishing operation. **Error correction** Check the input parameters for the finishing allowances and the machining strategy, and correct them if required.

Error number	Description
280-048B	**Error message** Machine datum could not be written **Cause of error** The value of the machine datum (MP_refPos) cannot be changed. The required change amount is greater than MP_maxModification/5, or MP_positionDiffRef/5. **Error correction** If you restore this data record the active machine kinematics might be inaccurate. Check the values and make the necessary adjustments manually. If necessary, inform your machine manufacturer.
280-048C	**Error message** Spindle for synchronization could not be ascertained **Cause of error** Could not determine the spindle to be synchronized. The spindle can be determined automatically only if exactly two spindles are configured in the system. The spindle to be synchronized must not be the active channel spindle. **Error correction** - If more than two spindles are configured, the spindle to be synchronized must be defined in the macro OEM_CYCLGEAR_PRE. - Contact your machine tool builder.
280-048D	**Error message** Function is not possible in the active operating mode **Cause of error** The programmed function is not possible in the active operating mode. **Error correction** For example, use FUNCTION MODE MILL or FUNCTION MODE TURN to activate the operating mode intended for the programmed function.
280-048E	**Error message** Oversize defined too large **Cause of error** The programmed allowance is greater than the entire machining depth. With gear teeth the machining depth corresponds to the tooth height: tooth height = 2 * module + trough-to-tip clearance **Error correction** Check the value of the programmed allowance.

Error number	Description
280-048F	**Error message** Number of teeth not defined **Cause of error** The number of teeth has not been defined for the active tool. The programmed machining operation requires the information about the number of teeth. **Error correction** In CUT column of the tool table, define the number of teeth.
280-0490	**Error message** Machining depth does not increase monotonously **Cause of error** A calculated machining depth does not increase monotonously. The programmed entries result in a machining depth that was already exceeded by a previous cut. **Error correction** Reduce either the first infeed or the number of infeeds. The last infeed must be smaller than the first one. Check the following entries: - First infeed Q586 - Last infeed Q587 - Number of infeeds Q584
280-0491	**Error message** Infeed does not decrease monotonously **Cause of error** A calculated infeed does not decrease strictly monotonously. The programmed entries result in at least one infeed that is greater than or equal to the previous one. **Error correction** Increase the number of infeeds or the first infeed. Reduce the last infeed. The last infeed must be smaller than the first one. Check the following entries: - First infeed Q586 - Last infeed Q587 - Number of infeeds Q584

Error number	Description
280-0492	**Error message** Tool radius not defined correctly **Cause of error** The physical tool radius (sum of R and DR from the tool table) is less than zero. If the physical tool radius equals zero, the selected machining cycle instead uses the programmed delta value DR. The effective tool radius (sum of R and DR from the tool table and the programmed delta value DR) is less than or equal to zero. **Error correction** Enter the correct radius of the tool in the tool table. A programmed delta value DR must not lead to an effective tool radius of less than or equal to zero.
280-0493	**Error message** Mode for retraction to clearance height not possible **Cause of error** The programmed mode for retraction to clearance height will be ignored for manual pre-positioning. **Error correction** When manually pre-positioning to the probing object, make sure that this movement is without collision.
280-0494	**Error message** Gear wheel definition incorrect **Cause of error** The definition of the tooth geometry is incomplete or contradicts itself. The module and number of teeth are necessary for the definition of a gear tooth system according to DIN 3990 (ISO 6336). Outside diameter and tooth height are optional entries. The outside diameter must be greater than the inside diameter. **Error correction** Enter the module and the number of teeth. If there are deviations from DIN 3990 (ISO 6336) then you can define the outside diameter and tooth height. Check the definition of the outside diameter and tooth height.

Error number	Description
280-0495	**Error message** Probing object contains different types of dimension definition **Cause of error** You didn't use the same type of dimension definition for all coordinates of an object to be probed. - For manual pre-positioning: Enter a question mark (?) at the beginning of a dimension definition in order to define manual pre-positioning. - For defining the actual position: Enter the at sign (@) after stating the nominal position in order to define the actual position. **Error correction** You must program the same type of dimension definition in the principal, secondary, and tool axes of an object to be probed. Correct the faulty dimension definition.
280-0496	**Error message** Dimension definition contains impermissible characters **Cause of error** A dimension definition contains impermissible characters. - There is more than one decimal separator in a value. - There are other characters after a dimension definition. - Impermissible characters were used. **Error correction** Correct the dimension definition.
280-0497	**Error message** Actual value in dimension definition faulty **Cause of error** The actual value is not indicated correctly in a dimension definition: - The preceding delimiter '@' is missing. - The definition of a value after the delimiter '@' is missing. The indicated Q parameter may not be used after the delimiter. **Error correction** Correct the definition of the actual value. You can only use Q1900 to Q1999 to transfer variable values.

Error number	Description
280-0498	**Error message** Starting point of hole too deep **Cause of error** The starting point Q379 of a hole is defined to be larger or equivalent to the total depth Q201 of the hole. **Error correction** Define the starting point to be within the specified hole depth.
280-0499	**Error message** Dimension def.: Nominal value missing for manual pre-positioning **Cause of error** When probing with manual pre-positioning, the nominal values are missing for all directions at one position. **Error correction** Define a nominal value for at least one direction. You should define a nominal value in at least the directions that you can specify exactly with the probing process. For manual pre-positioning, define the nominal value after the '?'.
280-049A	**Error message** A replacement tool is not available **Cause of error** The programmed tool is locked or the tool life has expired and no replacement tool is available. **Error correction** Check the columns TL, RT, CUR_TIME, and TIME2 of the programmed tool. If you programmed a tool number then the replacement tool is defined in the column RT. If you are using a tool name then define the same name for the replacement tool.
280-049B	**Error message** OEM macro is not defined **Cause of error** No macro is configured for this cycle. **Error correction** - In CfgSystemCycle, create a key with the name OEM_MACHSTAT_MEAS and store a macro - Contact your machine tool builder

Error number	Description
280-049C	**Error message** Measurement not possible with auxiliary axis **Cause of error** An OEM macro defined an axis to be measured that is not possible with this type of measurement. This measurement cannot be performed with auxiliary axes (PLC axes). **Error correction** - Change the type of measurement or the axis to be measured - Contact your machine tool builder
280-049D	**Error message** Start position not possible with modulo axis **Cause of error** The movement commanded in the OEM macro leads through the zero crossover of a modulo axis. **Error correction** - Pre-position the modulo axis such that the commanded movement does not lead through the zero crossover - Contact your machine tool builder
280-049E	**Error message** Function only possible if door is closed **Cause of error** The function you selected can be executed only if the guard doors are closed. **Error correction** Close the guard doors.
280-049F	**Error message** Number of possible records exceeded **Cause of error** Not enough memory is available in order to process the data. Cycle 453: Excessive number of measuring points. **Error correction** Reduce the number of records. Cycle 453: Reduce the number of rows in the compensation table (*.kco). Contact your machine tool builder.

Error number	Description
280-04A0	**Error message** Inconsistent machining plane due to axis angle with basic rot. **Cause of error** Das Schwenken der Bearbeitungsebene mit Achswinkeln in Kombination mit einer Grunddrehung führt zu einer inkonsistenten Bearbeitungsebene. Die Achswinkel stimmen nicht mit den Schwenkwinkeln überein. Das kann zu fehlerhaften Bearbeitungen führen. **Error correction** Vermeiden Sie die Kombination von Grunddrehung und Bearbeitungsebene schwenken mit Achswinkel.
280-04A1	**Error message** Transfer parameter contains an impermissible value **Cause of error** A transfer parameter from an OEM macro to the cycle is not in the permitted range. **Error correction** - Check the values transferred from the OEM macro to the cycle - Contact your machine manufacturer
280-04A2	**Error message** Tooth width RCUTS is defined too large **Cause of error** The tooth width RCUTS is defined too large for helical or reciprocating plunging in cycles 251 to 254. **Error correction** For helical or reciprocating plunging the tooth width must be less than the tool radius Correct the value for tooth width RCUTS in the tool table.
280-04A3	**Error message** Usable length LU of the tool is too small **Cause of error** The programmed machining depth is greater than the usable length of the tool. **Error correction** Check the usable length LU in the tool table. Correct the machining depth or use a different tool.

Error number	Description
280-04A4	**Error message** The defined chamfer is too large **Cause of error** The programmed chamfer is too large. **Error correction** Define a greater machining depth for the tip of the tool. Use a tool with a larger radius.
280-04A5	**Error message** Chamfer angle cannot be machined with the active tool **Cause of error** The programmed angle of the chamfer cannot be machined with the active tool. **Error correction** Check the value in input parameter Q354: The chamfer angle must be half of the point angle (T-ANGLE) of the tool. Enter the value 0 in Q354 in order to create a chamfer with half of the point angle (T-ANGLE) of the tool.
280-04A6	**Error message** The allowances do not define any stock removal **Cause of error** No stock removal is defined. The programmed allowance at the beginning of the machining operation is not greater than the allowance remaining at the end of the operation. **Error correction** Define the lateral allowance at the beginning of the machining operation (Q368) to be greater than the allowance at the end of the operation (Q14).

Error number	Description
280-04A7	**Error message** Spindle angle not unique **Cause of error** You tried to orient the tool spindle even though the relative position between the input and tool coordinate systems prevents an unambiguous determination of the spindle angle. During a probing operation the center offset (CAL_OF1 and CAL_OF2) defined for the touch probe cannot be taken into account correctly. **Error correction** Avoid a spindle orientation under these conditions: - Status of the transformations and position of tilting axes - Active turning operation Probing: - Exactly align the touch probe mechanically and do not define any values for the center offset (CAL_OF1 and CAL_OF2).
280-04A8	**Error message** Internal software error: wrong or faulty command **Cause of error** An internal software error has occurred. An unexpected or faulty command was received. **Error correction** Inform your service agency
280-04A9	**Error message** Probing procedure is not possible **Cause of error** The defined probing procedure cannot be performed with the active tool. **Error correction** Check whether the active tool is a touch probe. If the tool is not a touch probe, then you can apply the current coordinates with the actual position capture key.
280-04AA	**Error message** Type of the object to be probed is not possible **Cause of error** The type of the object selected to be probed cannot be probed in this situation. **Error correction** Select a different object to be probed. - Inform your service agency.

Error number	Description
280-04AB	**Error message** Input value not permitted **Cause of error** One of the input values is not in the valid range. **Error correction** Check and correct the input values.
280-04AC	**Error message** No data available for processing **Cause of error** No data found to be processed. **Error correction** Select the rows with the data you want to process. Enter nominal values for the processing.
280-04AD	**Error message** Object to be probed cannot be repeated **Cause of error** The object you selected to be probed cannot be re-probed. Changes to the basic rotation or the rotary-axis offset prevent correct calculation of the overall result. **Error correction** If necessary, re-probe all objects to be probed with the probing function.
280-04AE	**Error message** Limitation for island missing **Cause of error** In the definition of an island, the associated boundary is missing. **Error correction** Use Cycles 1281 or 1282 to define a boundary geometry for an island (Q650=1).
280-04AF	**Error message** Cannot process the data **Cause of error** The selected information cannot be used by this processing method. A basic rotation or an offset cannot be written to a datum table. A basic rotation cannot be written to a pallet preset. The combination of offset and shift (X,Y,Z) cannot be written to a pallet preset. **Error correction** Change the information selected. Use a different processing method.

Error number	Description
280-04B0	**Error message** Rotation of the tool coordinate system not permitted **Cause of error** An impermissible rotation of the tool coordinate system is in effect. This rotation can result in an improper motion during lift-off. **Error correction** Reset this rotation, for example with Cycle 801. Contact your machine tool builder.
280-04B1	**Error message** NC Start was ignored **Cause of error** NC Start was ignored since the current situation does not permit it. **Error correction** - Check the entries and correct them if necessary - Only press NC Start when the operational situation permits it
280-04B2	**Error message** Circle cannot be calculated **Cause of error** Could not calculate a circle from the measured points. The number of points or the distance between them is insufficient. **Error correction** Check the number and positions of the points for circle calculation.
280-04B3	**Error message** Probing of an extrusion not possible **Cause of error** An extrusion that cannot be probed was defined as probing object. An extrusion cannot be combined with a manual pre-positioning task for which a '?' is programmed in the position definition. **Error correction** Check your entries in Cycle 1493. Check the definition of the position of the probing object.

Error number	Description
280-04B4	**Error message** Nominal position not defined correctly **Cause of error** No nominal position is defined, or it contains tolerances. **Error correction** Enter a nominal position for all coordinate directions. Enter a tolerance along the surface-normal vector instead of for the nominal position.
280-04B5	**Error message** Retraction behavior not possible with multiple plunging **Cause of error** Only linear retraction is possible with multiple plunging. **Error correction** If necessary, adjust the input parameters Q462 Retraction mode and Q562 Multiple plunging.
280-04B6	**Error message** Basic rotation will be rescinded **Cause of error** A basic rotation is in effect in the active preset. In order for the sequence to work correctly, the called probing cycle requires that this basic rotation be rescinded. **Error correction** Check your entries. Clear this message and press NC-START in order to rescind the basic rotation and continue with the probing cycle. Or perform an internal stop in order to cancel the probing cycle and check your entries.
280-04B7	**Error message** Repeat last measurement? **Cause of error** The last measurement could not be performed correctly. **Error correction** Refer to the preceding messages concerning the last measurement. You can acknowledge these messages and then repeat the last measurement with NC Start.

Error number	Description
280-04B8	**Error message** Continue with next measurement? **Cause of error** This measurement cannot be performed correctly. **Error correction** Refer to the preceding messages concerning the measurement. Check the configuration for this measurement. You can acknowledge these messages and jump to the next measurement with NC Start.
280-04B9	**Error message** File not found **Cause of error** The specified file could not be found. **Error correction** Ensure that the specified file exists and that the given path is correct.
280-04BA	**Error message** The entire plunging depth is greater than the tooth height The entire plunging depth is less than the tooth height **Cause of error** The sum of the defined infeeds does not match the tooth height. Twice the tooth height results from the difference between the outside diameter and the inside diameter. If the total infeed is greater than the tooth height, the machining operation is not carried out. If the total infeed is less than the tooth height, the machining operation can still be carried out. **Error correction** Check the sum of all infeeds in the INFEED column, and correct them if necessary.
280-04BC	**Error message** File cannot be opened **Cause of error** Could not open the file specified in the cycle. **Error correction** Check whether the file is there, whether the path is correct, and whether the file is in a readable format.

Error number	Description
280-04BF	**Error message** Status of transformations for simultaneous turning is not correct **Cause of error** The status of the transformations is not correct for the requested simultaneous turning operation with a FreeTurn tool. TCPM must be activated before the cycle can run. Before the cycle was run, a transformation was activated that is not possible with this type of operating mode. **Error correction** Activate TCPM before calling the cycle. Check your corrections in the WPL-CS, such as: FUNCTION CORRDATA WPL.
280-04C0	**Error message** Preset cannot be modified after probe objects have been probed **Cause of error** The modifications to the preset are not possible. Modifications to the preset are only possible as long as no object has been probed yet. **Error correction** Discard objects that have already been probed by ending the manually selected probing function. Then you can make the changes to the preset.
280-04C1	**Error message** Tolerance does not match probing direction **Cause of error** The result of a probing motion is outside of the tolerance, and the direction of the probing motion does not match the defined tolerance. **Error correction** Check the programmed tolerances regarding the defined probing direction and the extrusion direction.
280-05DC	**Error message** Error in pallet management **Cause of error** Internal control error. **Error correction** Inform your service agency.

Error number	Description
280-05DD	**Error message** Error in pocket table **Cause of error** Error in pocket table: - Pockets or tools appear twice. - There is no value in column T for the spindle pocket. - The tool in the spindle is not in the tool table. - The TOOL_P symbol does not point to a pocket table or it is not set. - The pocket table is write-protected or it does not exist. **Error correction** - Correct the pocket table. - There is no value in column T for the spindle pocket. - The tool in the spindle is not in the tool table. - The TOOL_P symbol does not point to a pocket table or it is not set. - The pocket table is write-protected or it does not exist.
280-05DE	**Error message** Fixture not activated **Cause of error** Pallet changer: The started NC program belongs to a fixture that is not on the pallet. **Error correction** Activate the correct fixture.
280-05DF	**Error message** Wrong pallet **Cause of error** Pallet changer: The NC program that was started belongs to a pallet that is not at the machining position. The LOCATION column of the pallet table does not contain "MA". **Error correction** Change to the proper pallet (LOCATION == MA).
280-05E0	**Error message** Pallet line locked! **Cause of error** You attempted to run a locked pallet line. **Error correction** To resume program run, unlock the line or continue with the next line. If necessary, refer to your machine manual.

Error number	Description
280-05E1	**Error message** Datum table missing **Cause of error** You selected a datum table that does not exist in the control's NC memory. **Error correction** Select an existing datum table or make the desired table.
280-05E2	**Error message** Measuring probe not defined Touch probe not defined **Cause of error** - You called a touch probe that is not defined in the touch probe table. - The touch probe table is write-protected or it does not exist. **Error correction** - Add the missing touch probe to the touch probe table. - Create a touch probe table or cancel the write protection.
280-05E3	**Error message** Incorrect tool data **Cause of error** Incorrect tool data: - Tool appears twice. - TOOL symbol does not point to a tool table or it is not set. - The tool table is write-protected or it does not exist. - Tool table is locked because of the Test Run or Programming operating mode. **Error correction** - Correct the tool table. - Reassign the TOOL symbol or create an equivalent tool table. - Create a tool table or cancel the write protection. - Close the the Test Run or Programming operating mode.
280-05E4	**Error message** Tool number 0 not permitted **Cause of error** A tool definition with the number "0" is not permitted. **Error correction** Edit the part program.

Error number	Description
280-05E6	**Error message** No appropriate tool found **Cause of error** Automatic tool search: No suitable tool was found in the tool table. **Error correction** Check the tool table.
280-05E7	**Error message** Calculated tool number too large **Cause of error** - Calculation of a tool number from a Q parameter resulted in a value outside the permissible range of 0 to 32767. - You have called a tool number that is greater than the number of tools defined in the tool table. **Error correction** Edit the part program.
280-05E8	**Error message** Tool definition is missing Tool definition is missing **Cause of error** In a TOOL CALL (ISO: T..) you entered a tool number for which there is no definition in the program. **Error correction** Edit the part program.
280-05E9	**Error message** Tool number already assigned **Cause of error** You attempted to give a tool more than one definition. **Error correction** Edit the part program.
280-05EA	**Error message** Tool definition not permitted **Cause of error** You programmed a tool definition with radius or length (TOOL DEF, ISO: G99). **Error correction** - Delete the TOOL DEF block (G99 block). - Use the tool preselection without radius and length(TOOL DEF, ISO: G51).

Error number	Description
280-05EB	**Error message** TOOL DEF w/o length or radius **Cause of error** The definition of a tool (TOOL DEF, ISO: G99) is missing the value for tool length or tool radius. **Error correction** Complete the TOOL DEF block (G99 block).
280-05EC	**Error message** Max. tool age expired **Cause of error** The service life of the called tool has expired and you have not defined a replacement tool. **Error correction** Check the tool and, if necessary, exchange it or define a replacement tool.
280-05ED	**Error message** Tool locked **Cause of error** The tool was locked (e.g. after breakage). **Error correction** Check the tool and, if necessary, change it or unlock it in the tool table.
280-05EE	**Error message** Tool table is missing **Cause of error** Either you did not select a tool table or the selected table is not in the control's NC memory. **Error correction** Select an available tool table or create one.
280-05EF	**Error message** FN14_1519 **Cause of error** FN14_1519 **Error correction** FN14_1519
280-05F0	**Error message** Helical plunging not possible **Cause of error** Q366 = 1 **Error correction** Edit the part program

Error number	Description
280-05F1	**Error message** FN14_1521 **Cause of error** FN14_1521 **Error correction** FN14_1521
280-05F2	**Error message** No touch probe data **Cause of error** - No touch probe inserted - No tool axis active for the touch probe - Contradictory touch probe data **Error correction** - Insert the touch probe - Define the tool axis in the touch probe call - Check the touch probe data
280-05F3	**Error message** SQL command failed **Cause of error** An SQL command used in the cycle could not be executed. **Error correction** Inform your service agency.
280-05F4	**Error message** FN14_1524 **Cause of error** FN14_1524 **Error correction** FN14_1524
280-05F5	**Error message** FN14_1525 **Cause of error** FN14_1525 **Error correction** FN14_1525
280-05F6	**Error message** FN14_1526 **Cause of error** FN14_1526 **Error correction** FN14_1526

Error number	Description
280-05F7	**Error message** Error in pallet management **Cause of error** Error in pallet management: - The pallet table does not exist or is write-protected. - TARGET and FN17/18 ID510 NR22 are used simultaneous-ly in the PAL row. **Error correction** - Create a pallet table or cancel write-protection. - Do not use TARGET and FN17/18 ID510 NR22 simultane-ously in the PAL row.
280-05F8	**Error message** Error in preset table Error in preset table **Cause of error** The preset table is faulty. Possible causes: - The preset table is write-protected or it does not exist. - Line 0 does not exist. - The is no line with ACTNO = 1. **Error correction** - Please make the preset table or cancel the write protection - Enter the line 0 in the preset table - Set ACTNO in one line to zero
280-05F9	**Error message** Incorrect datum table **Cause of error** Incorrect datum table: - The datum table is write-protected or it does not exist. **Error correction** - Create a datum table or cancel the write protection.
280-05FA	**Error message** Tool change during mid-program startup not possible **Cause of error** Tool change not possible during mid-program startup. The active tool is not in the spindle for program run after mid-program startup. **Error correction** Please contact your machine tool builder.

Error number	Description
280-05FB	**Error message** Calibrate touch probe **Cause of error** You tried to automatically measure a tool although the tool touch probe is not yet calibrated. **Error correction** Calibrate the TT tool touch probe with the cycle TCH PROBE 30.
280-05FC	**Error message** Tool axis is missing **Cause of error** You called a fixed cycle without first activating a tool. **Error correction** Edit the NC program.
280-05FD	**Error message** CYCL DEF incomplete **Cause of error** - You deleted part of a cycle - You inserted other NC blocks within a cycle **Error correction** - Redefine the complete cycle - Delete NC blocks within a cycle
280-05FE	**Error message** TOOL.T: Enter number of teeth **Cause of error** Automatic tool measurement: Number of teeth not entered into tool table. **Error correction** Enter the number of teeth (CUT) into TOOL.T.
280-05FF	**Error message** Enter tool radius greater than 0 **Cause of error** You defined the cutter radius for the active tool in the tool table as less than or equal to 0. **Error correction** You can only measure a tool with a positive radius. Change the radius in the table.

Error number	Description
280-0600	**Error message** Tolerance in the parameter measureTolerance[1;2] is too low **Cause of error** The tolerance entered in the "measureTolerance1" parameter cannot be achieved during tool radius measurement with the TT. **Error correction** - Increase the permissible tolerance for finding tool teeth with spindle orientation in the "measureTolerance2" parameter - Reduce the positioning window of the spindle in the "posTolerance" parameter - Check whether a burr has formed on the probe contact. Remove the burr, if there has. - Exchange the tool touch probe, if required
280-0601	**Error message** Tool locked **Cause of error** The tool was locked (e.g. after breakage). **Error correction** Check the tool and, if necessary, change it or unlock it in the tool table.
280-0602	**Error message** Error in tool measurement configuration **Cause of error** The configuration of the tool measurement is incorrect or incomplete. **Error correction** Check the tool measurement configuration and edit or extend it, if required.
280-0603	**Error message** Tool measurement locked **Cause of error** Tool measurement is disabled. **Error correction** Check the tool measurement configuration and edit it, if required.

Error number	Description
280-0604	**Error message** Tool measurement: Functionality not yet implemented **Cause of error** The desired functionality is not implemented. - The tool axis is not the Z axis. (NC program: TOOL CALL) - Sum of "YL + DYL" is greater than +/– 5.0 mm. (Tool table) - Value of SPB-INSERT does not equal 0 degrees. (Tool table) - Value of ANGLE is greater than 90 degrees or sum of ANGLE + PANGLE is greater than or equal to 180 degrees. (Tool table) - The value TO is not supported for tool types. (Tool table) - The value of stylusAxis is not Z_Positive. (Configuration) **Error correction** Check the NC program, tool table, and tool measurement configuration and edit them as necessary.
280-0605	**Error message** Orientation not configured **Cause of error** - Your machine might not offer spindle orientation - Spindle orientation not possible **Error correction** - Refer to your machine manual! - Check the "spindleOrientMode" machine parameter and use the NC to enter -1 or the value of the M function for spindle orientation.
280-0606	**Error message** Arithmetical error **Cause of error** Internal calculations have resulted in a non-representable numerical value. **Error correction** Check the input values.
280-0607	**Error message** Cycle error **Cause of error** Internal control error **Error correction** Inform your service agency

Error number	Description
280-0608	**Error message** Tool broken **Cause of error** Automatic tool measurement: The breakage tolerance (LBREAK or RBREAK) from the tool table was exceeded. **Error correction** Check the tool and, if necessary, replace it.
280-0609	**Error message** Calibrate TT in tilted plane **Cause of error** You attempted to run a cycle for tool measurement while the tilted-plane function was active, although the touch probe was not calibrated in the tilted working plane. **Error correction** Run the calibration cycle 30 while the working plane is tilted.
280-060A	**Error message** Calibrate TT in non-tilted plane **Cause of error** You attempted to run a cycle for tool measurement, although the touch probe was last calibrated in a tilted working plane. **Error correction** Run the calibration cycle 30 when the working plane is not tilted.
280-060B	**Error message** TT not parallel to tool axis **Cause of error** You attempted to run a cycle for tool measurement although the touch probe axis is not parallel to the tool axis. **Error correction** Position the axes so that the touch probe axis and tool axis are parallel.
280-060C	**Error message** Tool index not allowed **Cause of error** You called a fixed cycle for tool measurement with a step drill. **Error correction**

Error number	Description
280-060D	**Error message** Turning tool incompletely defined Turning tool incompletely defined **Cause of error** - You called a turning tool that is not defined in the turning tool table. - The turning tool table is faulty or missing. **Error correction** - Add the missing tool to the turning tool table. - Create or correct the turning tool table.
280-060E	**Error message** Probing direction not in probe plane **Cause of error** You configured a probing direction that does not lie in the touch probe plane. **Error correction** Correct the machine parameter probingDirRadial
280-060F	**Error message** Unbalance detection failed **Cause of error** Fehler bei Unwuchterfassung aufgetreten **Error correction** Weitere Fehlermeldungen beachten
280-0610	**Error message** Excessive unbalance **Cause of error** Maximale Unwuchtamplitude überschritten **Error correction** Unwucht neu erfassen und kompensieren
280-0611	**Error message** Configuration for unbalance detection wrong **Cause of error** Die Konfiguration der Unwuchterfassung ist fehlerhaft oder unvollständig. **Error correction** Konfiguration der Unwuchterfassung überprüfen und ggf. anpassen oder erweitern.

Error number	Description
280-0612	**Error message** Radius of replacement tool not suitable **Cause of error** - During automatic exchange of a replacement tool (M101), the TNC did not find a suitable tool in the tool table. - The total radius R + DR of the replacement tool is greater and/or R2 + DR2 is less than the radius of the current tool when 3-D compensation is active. **Error correction** - Define a replacement tool with suitable radii. - Perhaps use M107 to deactivate monitoring of the tool radii.
280-0613	**Error message** Invalid tool axis programmed **Cause of error** You have programmed a tool axis other than Z. **Error correction** Edit the NC program.
280-0614	**Error message** Tool table locked **Cause of error** The tool file (TOOL.T) cannot be edited while the TNC is executing a tool call. Pressing the EDIT ON/OFF soft key provokes this error message. **Error correction** Exit the mode by selecting the "EDIT ON/OFF" soft key. Then acknowledge the message and resume program run with NC Start.
280-0615	**Error message** Unbalance calculation failed **Cause of error** An error occurred while calculating the unbalance. The entered value is not in the unbalance table. **Error correction** - Modify the entered value - Expand the unbalance table
280-0617	**Error message** Traverse mode for retraction not possible **Cause of error** The “tilted system” and “tool axis” traverse modes are not possible because of the machine configuration. **Error correction** Select the “machine axes” or “thread” traverse modes and repeat the retraction.

Error number	Description
280-0618	**Error message** Tool life expired **Cause of error** The remaining tool life is not enough for the precalculated machining time. - The service life of the called tool is insufficient and you haven’t defined a sister tool. - The tool-usage file is not available or not up to date. **Error correction** - The tool is to be used anyway: acknowledge the message and continue the NC program with NC start. - The tool is not to be used: cancel the NC program with an INTERNAL STOP. o Check the tool and, if necessary, exchange it or define a replacement tool. o Create or update a tool-usage file. Run the desired program in the Test Run mode of operation. Ensure that creation of a tool usage file is activated in the configuration.
280-0619	**Error message** Tool life expired **Cause of error** The remaining tool life is not enough for the precalculated machining time. - The service life of the called tool has expired and you have not defined a sister tool. - The tool usage file is not available or not up to date. **Error correction** - Check the tool and, if necessary, exchange it or define a sister tool. - Create or update a tool usage file. - Run the desired program in the Test Run mode of operation. - Ensure that the tool usage file is activated in the configuration.
280-061A	**Error message** Feed rate limiting has been canceled **Cause of error** - Feed rate limiting was canceled by the operator. - Reactivation of the feed rate limiting is not possible in the Retraction operating mode. **Error correction** - Use the feed rate potentiometer F to limit the feed rate. - Be very careful when moving the axes.

Error number	Description
280-061B	**Error message** Pallet line with completed part **Cause of error** The pallet line under the cursor is marked as a completed part and can therefore no longer be executed. **Error correction** Select a pallet line in which a workpiece blank or an incomplete part is entered.
280-061C	**Error message** Access to pallet preset table failed **Cause of error** Access to pallet preset table was not possible. The pallet preset table might not exist or is faulty. **Error correction** Inform your service agency.
280-061D	**Error message** Automatic continuation of pallet machining not possible **Cause of error** Automatic continuation of pallet machining not possible. **Error correction** - Pallet machining was not continued by the OEM cycle. - Check the entry for the OEM macro in OEM_PAL_RESUMPTION. - Inform your service agency.
280-061F	**Error message** Thread-cutting process was interrupted **Cause of error** Automatic continuation of pallet machining not possible. **Error correction** - Note any further error messages. - Correct the cause of error and repeat the operation. - Inform your service agency.
280-0620	**Error message** Pallet table locked **Cause of error** An editor locked any further execution of the pallet table **Error correction** - Exit the editing or input mode - Then acknowledge the message and resume the pallet machining with NC start.

Error number	Description
280-0621	**Error message** Tool not defined completely **Cause of error** You called a tool that is not completely defined in the tool table: - There is no value for the tool radius and/or length. **Error correction** - Check and complete the tool entries. - Use another tool.
280-0622	**Error message** Row does not exist in preset table **Cause of error** Could not activate the programmed preset. The given line does not exist in the preset table. **Error correction** - Check the preset table - Add the given line to the preset table.
280-0623	**Error message** Row does not exist in pallet preset table **Cause of error** Could not activate the programmed number of the pallet preset. The given line does not exist in the pallet preset table. **Error correction** - Check the pallet preset table - Add the given line to the pallet preset table.
280-06A4	**Error message** Camera not reacting Camera not responding **Cause of error** The image processing cycle cannot communicate with the camera or does not answer. **Error correction** Restart the NC software or remove the image processing cycle from the NC program. - Inform your service agency.
280-06A6	**Error message** File for camera position does not exist **Cause of error** The control cannot move the camera to the desired position because the table with the position data is missing. **Error correction** Inform your service machine tool builder.

Error number	Description
280-06A7	**Error message** Line does not exist in the position table **Cause of error** The control cannot move the camera to the desired position because the desired line number is missing in the table for position data. **Error correction** Inform your machine tool builder
280-06A8	**Error message** Communication with camera not possible **Cause of error** The image processing cycle cannot communicate with the camera because the internal data structure for communication is missing. **Error correction** - Inform your service agency.
280-06A9	**Error message** Camera provides no images **Cause of error** No live image was received from the camera. **Error correction** - Test whether the live image is shown correctly in the Manual operating mode - If this is not the case, restart the control - If both actions don't help, contact your service agency
280-06AA	**Error message** A name for the monitoring point is missing **Cause of error** The monitoring point has no name **Error correction** Enter a name for the monitoring point in the image processing cycle
280-06AB	**Error message** Not allowed to call an unbalance cycle in milling mode **Cause of error** A unbalance cycle cannot be started in milling mode **Error correction** Start the unbalance cycle in turning mode

Error number	Description
280-07D0	**Error message** Invalid error consequence **Cause of error** Invalid error consequence **Error correction** Internal error
280-07D1	**Error message** No space released **Cause of error** No location released for inserting the tool. On the wheel head there are no are no locations released for inserting the grinding wheel. Error correction: Release locations for inserting the tool (table: WHEEL.PGW). **Error correction** Release locations for inserting the tool (table: WHEEL.PGW).
280-07D2	**Error message** No valid physical space **Cause of error** No valid, physical location on wheel head. Selected location on wheel head is invalid. Location number is greater than 9. **Error correction** Select valid location (0...9).
280-07D3	**Error message** Space not released **Cause of error** Location for inserting the tool not released. The selected location on the wheel head has not been released and therefore cannot be accept a grinding wheel. **Error correction** Select another location for insertion.
280-07D4	**Error message** Tool is already inserted **Cause of error** The tool to be inserted is already clamped in another location. **Error correction** Select another tool (other tool number) or remove the tool from another location (not only physical, but also logical).

Error number	Description
280-07D5	**Error message** No space released **Cause of error** No released location found. No other vacant location was found on the wheel head. **Error correction** Remove tool from another location and use the set-up function to release the location.
280-07D6	**Error message** No identical space **Cause of error** No released, identical location found. There is no other logical location on the wheel head with the same physical position. **Error correction** Remove tool from another location and use the set-up function to release the location.
280-07D7	**Error message** No further tool found **Cause of error** No other tool found that meets the search criteria. **Error correction** Adapt the search criteria.
280-07D8	**Error message** No further tool found **Cause of error** No tool found that meets the search criteria. **Error correction** Adapt the search criteria.
280-07D9	**Error message** Invalid tool type **Cause of error** An unsupported tool type was selected, or the tool type is not permitted together with a function. **Error correction** Check the tool type.

Error number	Description
280-07DA	**Error message** Tool type not allowed **Cause of error** Tool type not allowed, not released. In the set-up function a tool was selected that is not supported at present. **Error correction** Select another tool type.
280-07DB	**Error message** Data not loaded **Cause of error** Data were not accepted because the tool number was changed. In the set-up function, the number (reference) of the current tool was changed. Because of this, all edited data were discarded and the data of the tool with the new number were loaded. **Error correction** None
280-07DC	**Error message** No entry in table **Cause of error** No entry in table / Access failed. An attempt to access a table failed. It could be that the desired entry is missing or the format of a column doesn't fit. **Error correction** Check the values.
280-07DD	**Error message** Invalid parameter value **Cause of error** Invalid parameter value. A parameter has an invalid value. **Error correction** Check the values of the parameters.
280-07DE	**Error message** Invalid command **Cause of error** Invalid command. An invalid command (FN19 command) was programmed to the PLC. **Error correction** Check the command / Check the PLC program.

Error number	Description
280-07DF	**Error message** Error in table access **Cause of error** Error in table access (no handle). An attempt to access a table failed. It could be that the desired entry is missing, the format of a column doesn't fit, the table doesn't exist, etc. **Error correction** Check the values.
280-07E0	**Error message** No grinding wheel **Cause of error** Current tool is not a grinding wheel. It is expected that the current tool be a grinding wheel, which is not the case. To find the location of a dresser, for example, the current tool must be a grinding wheel. **Error correction** Exchange the grinding wheel.
280-07E1	**Error message** Invalid tool number **Cause of error** Invalid tool number. The tool number is out of range. **Error correction** Enter the correct number (1...99).
280-07E2	**Error message** Invalid alignment **Cause of error** Invalid dresser alignment. The alignment of a dresser doesn't fit the selected grinding wheel edge. **Error correction** Select another wheel edge or another dresser alignment.
280-07E3	**Error message** No dresser defined **Cause of error** No dresser defined. A function expects that a dresser has been defined/programmed, which is not the case. **Error correction** Define/program a dresser.

Error number	Description
280-07E4	**Error message** Dresser not defined **Cause of error** Dresser not captured for this wheel. You tried to work with a dresser whose location hasn't been found for the current wheel. **Error correction** Capture/set-up the dresser.
280-07E5	**Error message** Not a valid wheel type **Cause of error** Not a valid wheel type. Invalid wheel type programmed. **Error correction** Select the correct wheel type.
280-07E6	**Error message** Ref. of dresser to wheel edge **Cause of error** Reference of dresser to wheel edge is incorrect. The dresser is being used in reference to another wheel edge, as if it were captured/set-up. **Error correction** Repeat the capture/setup of the dressing attachment.
280-07E7	**Error message** Relieved wheel not possible **Cause of error** Wheel side: Relieved wheel not possible. The length of the relief is not possible, or the combination with other parameters of the wheel side is not possible. **Error correction** Check the parameters of the wheel side.
280-07E8	**Error message** Chamfer width missing **Cause of error** Wheel side: Chamfer width missing. A chamfer width is expected but was not programmed. **Error correction** Check the parameters of the wheel side.

Error number	Description
280-07E9	**Error message** Chamfer greater than side len. X **Cause of error** Wheel side: Chamfer greater than side length X. **Error correction** Check the parameters of the wheel side.
280-07EA	**Error message** Angle of the relief is incorrect **Cause of error** Wheel side: Angle of the relief is incorrect. The angle of the relief has an invalid value. **Error correction** Check the parameters of the wheel side.
280-07EB	**Error message** FN14_2027 **Cause of error** FN14_2027 **Error correction** FN14_2027
280-07EC	**Error message** FN14_2028 **Cause of error** FN14_2028 **Error correction** FN14_2028
280-07ED	**Error message** FN14_2029 **Cause of error** FN14_2029 **Error correction** FN14_2029
280-07EE	**Error message** Dressing strategy: corner radius not permitted **Cause of error** If a corner radius (RV, RV1, RV2) is defined, then a dressing strategy must be selected that dresses the diameter and the side at the same time. **Error correction** Select a different dressing cycle, or set corner radius to 0

Error number	Description
280-07EE	**Error message** FN14_2030 **Cause of error** FN14_2030 **Error correction** FN14_2030
280-07EF	**Error message** Dressing strategy: wheel edge not supported **Cause of error** The combination of dressing cycle and active wheel edge is not allowed **Error correction** Activate a different wheel edge or select a different dressing cycle
280-07EF	**Error message** FN14_2031 **Cause of error** FN14_2031 **Error correction** FN14_2031
280-07F0	**Error message** Selected dressing strategy is not supported **Cause of error** A reciprocating strategy was programmed although this is not supported. A reciprocating strategy can be used only if the dressing movement consists of a straight line. A “Special grinding point” type of grinding wheel cannot be used with the “reciprocating” strategy. **Error correction** Select a different dressing strategy
280-07F0	**Error message** FN14_2032 **Cause of error** FN14_2032 **Error correction** FN14_2032

Error number	Description
280-07F1	**Error message** Dressing mode already active, tool not allowed **Cause of error** If dressing mode (FUNCTION DRESS BEGIN) is activated before the dressing cycle, then no tool may be programmed in the dressing cycle. **Error correction** - Clear the tool number/name - Remove FUNCTION DRESS BEGIN before the dressing cycle
280-07F1	**Error message** FN14_2033 **Cause of error** FN14_2033 **Error correction** FN14_2033
280-07F2	**Error message** Type of grinding wheel not allowed, not approved **Cause of error** The dressing cycle is not suitable for the selected type of grinding wheel, or has not been released yet. **Error correction** Select a different dressing cycle
280-07F2	**Error message** FN14_2034 **Cause of error** FN14_2034 **Error correction** FN14_2034
280-07F3	**Error message** Tool is not a dressing wheel or roll **Cause of error** A relationship between the cutting speeds was programmed even though the dressing tool is neither a dressing wheel nor a roll. **Error correction** - Change the type of dressing tool - Either do not program the relationship between the cutting speeds, or set it to 0

Error number	Description
280-07F3	**Error message** FN14_2035 **Cause of error** FN14_2035 **Error correction** FN14_2035
280-07F4	**Error message** FN14_2036 **Cause of error** FN14_2036 **Error correction** FN14_2036
280-07F5	**Error message** FN14_2037 **Cause of error** FN14_2037 **Error correction** FN14_2037
280-07F6	**Error message** FN14_2038 **Cause of error** FN14_2038 **Error correction** FN14_2038
280-07F7	**Error message** FN14_2039 **Cause of error** FN14_2039 **Error correction** FN14_2039
280-07F8	**Error message** FN14_2040 **Cause of error** FN14_2040 **Error correction** FN14_2040

Error number	Description
280-07F9	**Error message** FN14_2041 **Cause of error** FN14_2041 **Error correction** FN14_2041
280-07FA	**Error message** FN14_2042 **Cause of error** FN14_2042 **Error correction** FN14_2042
280-07FB	**Error message** FN14_2043 **Cause of error** FN14_2043 **Error correction** FN14_2043
280-07FC	**Error message** FN14_2044 **Cause of error** FN14_2044 **Error correction** FN14_2044
280-07FD	**Error message** FN14_2045 **Cause of error** FN14_2045 **Error correction** FN14_2045
280-07FE	**Error message** FN14_2046 **Cause of error** FN14_2046 **Error correction** FN14_2046

Error number	Description
280-07FF	**Error message** FN14_2047 **Cause of error** FN14_2047 **Error correction** FN14_2047
280-0800	**Error message** FN14_2048 **Cause of error** FN14_2048 **Error correction** FN14_2048
280-0801	**Error message** FN14_2049 **Cause of error** FN14_2049 **Error correction** FN14_2049
280-0834	**Error message** Infeed not defined **Cause of error** Infeed not defined. The definition command for this infeed was not programmed. **Error correction** Define/program the infeed.
280-0835	**Error message** Infeed direction not defined **Cause of error** Infeed direction not defined. The infeed direction is unknown. That means that the start position of the infeed is identical with the end position, and no infeed direction is known from a previous command. **Error correction** At least in the first infeed command, program an end position unequal to the starting position.

Error number	Description
280-0836	**Error message** Undersize **Cause of error** Undersize. During grinding with a dimensional control, the control has already responded at the start of the command. This means that the diameter to be ground already has a finishing dimension or undersize. If more than one cycle is programmed in sequence with a dimensional control, it can be normal for the rough size to be too small. However, at least the last cycle must be free of such error. **Error correction** Check the workpiece / Check the setting of the dimensional control.
280-0837	**Error message** FN14_2103 **Cause of error** FN14_2103 **Error correction** FN14_2103
280-0838	**Error message** Oversize **Cause of error** Oversize. During grinding with a dimensional control, the control has not responded. This means that the diameter to be ground was not reached. The workpiece has an oversize. **Error correction** Check the workpiece / Check the setting of the dimensional control.
280-0839	**Error message** FN14_2105 **Cause of error** FN14_2105 **Error correction** FN14_2105

Error number	Description
280-083A	**Error message** Signal already active at start **Cause of error** Signal already active at start. During grinding with an external signal (probe etc.) the signal responded before the movement was started. **Error correction** Compare the workpiece dimensions with the program and correct the program if necessary.
280-083B	**Error message** FN14_2107 **Cause of error** FN14_2107 **Error correction** FN14_2107
280-083C	**Error message** Signal has not responded **Cause of error** Signal has not responded. During grinding with an external signal (probe etc.) the signal did not respond during the movement. **Error correction** Compare the workpiece dimensions with the program and correct the program if necessary.
280-083D	**Error message** FN14_2109 **Cause of error** FN14_2109 **Error correction** FN14_2109
280-083E	**Error message** B axis in wrong position **Cause of error** B axis in wrong position. For a function, the B axis was expected to be in a defined position. The B axis is not in this position. **Error correction** Check the position of the B axis.

Error number	Description
280-083F	**Error message** No grinding wheel **Cause of error** Tool to be dressed is not a grinding wheel. Only grinding wheels can be dressed. **Error correction** Check the tool type.
280-0840	**Error message** Dressing location not released **Cause of error** Dressing location has not been not released. A selected dressing location has not been not released. **Error correction** Select another dresser location.
280-0841	**Error message** Wheel edges not released **Cause of error** Backward grinding (edges 4-6) not released. The edges 4 to 6 are not released for the selected location. This means that "backward" grinding is not allowed. **Error correction** Select a wheel edge in front (1 to 3).
280-0842	**Error message** Wheel location not occupied **Cause of error** Location on wheelhead not occupied. There is no tool at the desired location. **Error correction** Select another location or tool.
280-0843	**Error message** Wheel location not released **Cause of error** Location on wheelhead not released. No tool is allowed at the desired location. It is locked against occupation. **Error correction** Select another location, or release this one for occupation.

Error number	Description
280-0844	**Error message** Dressing location not occupied **Cause of error** Dresser location is not occupied. The desired dresser attachment location is not occupied. **Error correction** Select another location or place a dressing attachment in the location.
280-0845	**Error message** T-call parameter invalid **Cause of error** Parameter during tool call is out of valid range. **Error correction** Check the tool call.
280-0846	**Error message** Wheel settings not defined **Cause of error** The settings of the grinding wheel are not defined. For certain setup functions it is expected that for a grinding wheel the settings (position of the head) have been defined/ set up. **Error correction** Define / set up the settings.
280-0847	**Error message** Wheel data not defined **Cause of error** Wheel data not captured (diameter, width, etc.). For certain setup functions it is expected that for one grinding wheel the settings (position of the head) have been defined/set up. **Error correction** Capture/set up the grinding wheel data.
280-0848	**Error message** Wheel not inserted **Cause of error** The grinding wheel is not mounted. Either no grinding wheel was mounted (logical wheel location number = 0 or wheel number = 0) or you attempted to work with a wheel other than the one that was mounted. **Error correction** Use tool call to mount the wheel.

Error number	Description
280-0849	**Error message** Wheel location changed **Cause of error** Wheel pocket changed. When the dresser was captured, the grinding wheel was at another location. **Error correction** Repeat the capture/setup of the dressing attachment.
280-084A	**Error message** Dressing location changed **Cause of error** Dressing location changed. When the wheel was evaluated, the dressing attachment was at another location. **Error correction** Repeat the evaluation/setup of the dressing attachment.
280-084C	**Error message** Feed rate not programmed **Cause of error** Feed rate not programmed.No feed rate was programmed, or it was programmed as zero. **Error correction** Program the feed rate for a velocity other than zero.
280-084D	**Error message** Wheel missing **Cause of error** Wheel missing. You tried to work with a location that is not occupied by a grinding wheel. **Error correction** Occupy the location with a grinding wheel.
280-084E	**Error message** No valid tool selected **Cause of error** No valid tool selected. You tried to work with a tool that is not allowed for the current operation, or no tool is even selected. **Error correction** Select a valid tool.

Error number	Description
280-084F	**Error message** FN14_2127 **Cause of error** FN14_2127 **Error correction** FN14_2127
280-0850	**Error message** Not a valid dresser type **Cause of error** Not a valid dresser type. You tried to work with a dresser that is not allowed for the current operation, or the dresser type is not defined. **Error correction** Define/check the dresser type.
280-0852	**Error message** No event programmed **Cause of error** No event programmed. You tried to run a function for which an event (touch probe, dimensional control, etc.) has to be defined, but no event is defined. **Error correction** Define/program the event.
280-0853	**Error message** Invalid event programmed **Cause of error** Invalid event programmed.An event was program that is not or not yet supported, or the event does not match the function. **Error correction** Program another event.
280-0854	**Error message** FN14_2132 **Cause of error** FN14_2132 **Error correction** FN14_2132

Error number	Description
280-0855	**Error message** Event happened before movement **Cause of error** Event happened before movement. An event has already happened before the associated movement was started. **Error correction** Check the program. Depending on the event, however, this may be correct behavior.
280-0856	**Error message** FN14_2134 **Cause of error** FN14_2134 **Error correction** FN14_2134
280-0857	**Error message** FN14_2135 **Cause of error** FN14_2135 **Error correction** FN14_2135
280-0858	**Error message** Event did not happen **Cause of error** The event did not happen. A movement with an event was programmed and the movement was completed without the event occurring. **Error correction** Check the program. Depending on the event, however, this may be correct behavior.
280-0859	**Error message** FN14_2137 **Cause of error** FN14_2137 **Error correction** FN14_2137

Error number	Description
280-085A	**Error message** No swing stroke **Cause of error** No swing stroke programmed. In a swing grinding operation, no swing stroke was programmed, or the stroke was programmed as zero. **Error correction** Check the swing cycle.
280-085B	**Error message** Jig grinding, reciprocating stroke: tool axis not allowed **Cause of error** The current tool axis is not supported by reciprocation cycle 1000 **Error correction** Reciprocation cycle 1000 is possible only with X, Y, or Z as tool axis
280-085B	**Error message** FN14_2139 **Cause of error** FN14_2139 **Error correction** FN14_2139
280-085C	**Error message** Jig grinding: reciprocating stroke already stopped **Cause of error** A reciprocation stop (cycle 1002) was programmed even though the reciprocation movement has already stopped. **Error correction** Check the NC program Mid-program startup, a change of operating mode, and other actions stop an active reciprocation movement
280-085C	**Error message** FN14_2140 **Cause of error** FN14_2140 **Error correction** FN14_2140

Error number	Description
280-085D	**Error message** Jig grinding: reciprocating stroke already defined **Cause of error** A reciprocation cycle (cycle 1000) was defined even though a reciprocation cycle is already active. **Error correction** Clear the previous reciprocation cycle definition (cycle 1002) before defining a new reciprocation cycle.
280-085D	**Error message** FN14_2141 **Cause of error** FN14_2141 **Error correction** FN14_2141
280-085E	**Error message** Immediate stop only permitted if reciprocation def. gets deleted **Cause of error** The parameter combination "Immediate stop" and "Do not delete reciprocation definition" is not allowed. **Error correction** Check the combination of parameter values, and correct as necessary
280-085E	**Error message** FN14_2142 **Cause of error** FN14_2142 **Error correction** FN14_2142
280-085F	**Error message** FN14_2143 **Cause of error** FN14_2143 **Error correction** FN14_2143
280-0860	**Error message** FN14_2144 **Cause of error** FN14_2144 **Error correction** FN14_2144

Error number	Description
280-0861	**Error message** FN14_2145 **Cause of error** FN14_2145 **Error correction** FN14_2145
280-0862	**Error message** FN14_2146 **Cause of error** FN14_2146 **Error correction** FN14_2146
280-0863	**Error message** FN14_2147 **Cause of error** FN14_2147 **Error correction** FN14_2147
280-0864	**Error message** FN14_2148 **Cause of error** FN14_2148 **Error correction** FN14_2148
280-0865	**Error message** FN14_2149 **Cause of error** FN14_2149 **Error correction** FN14_2149
280-0866	**Error message** Wrong axis programmed **Cause of error** An axis was programmed that is not allowed for the present function. **Error correction** Check the program.

Error number	Description
280-0867	**Error message** No axis programmed **Cause of error** No axis programmed. In a function that needs at least one programmed axis, no axis was programmed. **Error correction** Check the program.
280-0868	**Error message** M command not allowed **Cause of error** M command not allowed. An M command was programmed that is invalid or is not allowed at this time. **Error correction** Check the program.
280-0869	**Error message** FN14_2153 **Cause of error** FN14_2153 **Error correction** FN14_2153
280-086A	**Error message** FN14_2154 **Cause of error** FN14_2154 **Error correction** FN14_2154
280-086B	**Error message** FN14_2155 **Cause of error** FN14_2155 **Error correction** FN14_2155
280-086C	**Error message** FN14_2156 **Cause of error** FN14_2156 **Error correction** FN14_2156

Error number	Description
280-086D	**Error message** FN14_2157 **Cause of error** FN14_2157 **Error correction** FN14_2157
280-086E	**Error message** FN14_2158 **Cause of error** FN14_2158 **Error correction** FN14_2158
280-086F	**Error message** FN14_2159 **Cause of error** FN14_2159 **Error correction** FN14_2159
280-0870	**Error message** Pitch <= 0 **Cause of error** Thread grinding: Pitch <= 0. For thread grinding, the pitch must be greater than 0. **Error correction** Correct the parameter.
280-0871	**Error message** Rotational speed = 0 **Cause of error** Thread grinding: Rotational speed = 0. For thread grinding, the rotational speed must not be 0. **Error correction** Correct the parameter.
280-0872	**Error message** Cutting length = 0 **Cause of error** Thread plunge grinding: Cutting length = 0. For thread plunging, the cutting length must not be 0. **Error correction** Correct the parameter.

Error number	Description
280-0873	**Error message** Velocity = 0 **Cause of error** Thread plunge grinding: Velocity Ve, Vm or Vk = 0. For thread plunging, none of the three velocities can be zero. **Error correction** Correct the parameter.
280-0874	**Error message** Signs differ **Cause of error** Thread plunge grinding: Different signs for E, M and K. For thread plunge grinding, the algebraic signs of the parameters E, M and K must be identical. **Error correction** Correct the parameter.
280-0875	**Error message** Pitch = 0 **Cause of error** Thread plunge grinding: Thread depth = 0. For thread plunging, the thread depth must not be 0. **Error correction** Correct the parameter.
280-0876	**Error message** FN14_2166 **Cause of error** FN14_2166 **Error correction** FN14_2166
280-0877	**Error message** FN14_2167 **Cause of error** FN14_2167 **Error correction** FN14_2167
280-0878	**Error message** FN14_2168 **Cause of error** FN14_2168 **Error correction** FN14_2168

Error number	Description
280-0879	**Error message** FN14_2169 **Cause of error** FN14_2169 **Error correction** FN14_2169
280-087A	**Error message** FN14_2170 **Cause of error** FN14_2170 **Error correction** FN14_2170
280-087B	**Error message** FN14_2171 **Cause of error** FN14_2171 **Error correction** FN14_2171
280-087C	**Error message** FN14_2172 **Cause of error** FN14_2172 **Error correction** FN14_2172
280-087D	**Error message** FN14_2173 **Cause of error** FN14_2173 **Error correction** FN14_2173
280-087E	**Error message** FN14_2174 **Cause of error** FN14_2174 **Error correction** FN14_2174

Error number	Description
280-087F	**Error message** FN14_2175 **Cause of error** FN14_2175 **Error correction** FN14_2175
280-0880	**Error message** FN14_2176 **Cause of error** FN14_2176 **Error correction** FN14_2176
280-0881	**Error message** FN14_2177 **Cause of error** FN14_2177 **Error correction** FN14_2177
280-0882	**Error message** FN14_2178 **Cause of error** FN14_2178 **Error correction** FN14_2178
280-0883	**Error message** FN14_2179 **Cause of error** FN14_2179 **Error correction** FN14_2179
280-0884	**Error message** Block scan over probe functions not allowed **Cause of error** Block scan over probe functions not allowed. A block scan was started over a block with a probe function. **Error correction** Run the block without block scan. Touch probe functions cannot be run in block scan.

Error number	Description
280-0885	**Error message** Command not allowed during block scan **Cause of error** Command not allowed during block scan. A command or cycle cannot be run in block scan. **Error correction** Run the block without block scan. Some commands cannot be run in block scan.
280-0886	**Error message** Command not executed due to block scan **Cause of error** Command not executed due to block scan. A command or cycle was not run due to in block scan. **Error correction** None
280-0887	**Error message** Measuring function not executed due to block scan **Cause of error** Measuring function not executed due to block scan. A command or cycle containing a measuring function was not run due to block scan. **Error correction** None
280-0888	**Error message** Elimination of air grinding was not executed due to block scan **Cause of error** Elimination of air grinding was not executed due to block scan. A command or cycle containing the "Eliminate air grinding" function was not run due to block scan. **Error correction** None
280-0889	**Error message** Block scan is not possible on this block **Cause of error** Block scan is not possible on this block. The control does not support a block scan on the selected block. **Error correction** Select a block scan on another block.

Error number	Description
280-088A	**Error message** FN14_2186 **Cause of error** FN14_2186 **Error correction** FN14_2186
280-088B	**Error message** FN14_2187 **Cause of error** FN14_2187 **Error correction** FN14_2187
280-088C	**Error message** FN14_2188 **Cause of error** FN14_2188 **Error correction** FN14_2188
280-088D	**Error message** FN14_2189 **Cause of error** FN14_2189 **Error correction** FN14_2189
280-088E	**Error message** Command not allowed in the simulation **Cause of error** Command not allowed in simulation. The command is not supported by the control in the simulation. **Error correction** Do not use the command in the simulation.
280-088F	**Error message** FN14_2191 **Cause of error** FN14_2191 **Error correction** FN14_2191

Error number	Description
280-0890	**Error message** FN14_2192 **Cause of error** FN14_2192 **Error correction** FN14_2192
280-0891	**Error message** FN14_2193 **Cause of error** FN14_2193 **Error correction** FN14_2193
280-0892	**Error message** FN14_2194 **Cause of error** FN14_2194 **Error correction** FN14_2194
280-0893	**Error message** FN14_2195 **Cause of error** FN14_2195 **Error correction** FN14_2195
280-0894	**Error message** FN14_2196 **Cause of error** FN14_2196 **Error correction** FN14_2196
280-0895	**Error message** FN14_2197 **Cause of error** FN14_2197 **Error correction** FN14_2197

Error number	Description
280-0896	**Error message** FN14_2198 **Cause of error** FN14_2198 **Error correction** FN14_2198
280-0897	**Error message** FN14_2199 **Cause of error** FN14_2199 **Error correction** FN14_2199
280-0898	**Error message** No safety clearance at diameter **Cause of error** No safety clearance at diameter. No safety clearance was programmed at the diameter of the grinding wheel. When the dresser approaches the grinding wheel, it moves toward the wheel edge up to a safety clearance. To ensure that the diamond does not touch the wheel at this position, the safety clearance must be at least as large as half the dresser width. **Error correction** Define a safety clearance at the diameter.
280-0899	**Error message** No safety clearance on the outside **Cause of error** No safety clearance on the outside. No safety clearance was programmed at the outer side of the grinding wheel. When the dresser approaches the grinding wheel, it moves toward the wheel edge up to a safety clearance. To ensure that the diamond does not touch the wheel at this position, the safety clearance must be at least as large as half the dresser width. **Error correction** Define a safety clearance on the outer side.

Error number	Description
280-089A	**Error message** No safety clearance on the inside **Cause of error** No safety clearance on the inner side. No safety clearance was programmed at the inner side of the grinding wheel. When the dresser approaches the grinding wheel, it moves toward the wheel edge up to a safety clearance. To ensure that the diamond does not touch the wheel at this position, the safety clearance must be at least as large as half the dresser width. **Error correction** Define a safety clearance on the inner side.
280-089B	**Error message** Dresser too wide **Cause of error** Dresser too wide (safety clearance too small). When the dresser approaches the grinding wheel, it moves toward the wheel edge up to a safety clearance. To ensure that the diamond does not touch the wheel at this position, the safety clearance must be at least as large as half the dresser width. **Error correction** Check the safety clearances.
280-089C	**Error message** Insufficient diameter **Cause of error** Wheel diameter is too small. The wheel has less than the minimum permissible diameter. The minimum diameter requirement might have been violated during dressing or in a corresponding entry during setup. **Error correction** Correct the entry or adjust the minimum wheel diameter. You might have to insert another wheel.
280-089D	**Error message** Insufficient width **Cause of error** Wheel width is too small. The wheel has less than the minimum permissible width. The minimum width requirement might have been violated during dressing or in a corresponding entry during setup. **Error correction** Correct the entry or adjust the minimum wheel width. You might have to insert another wheel.

Error number	Description
280-089E	**Error message** Outer side of wheel incorrect **Cause of error** Outer side of wheel incorrect. Incorrect values or incorrect combination of parameters that define the outer side of the wheel. **Error correction** Check the parameters of the outer side of the wheel.
280-089F	**Error message** Inner side of wheel incorrect **Cause of error** Inner side of wheel incorrect. Incorrect values or incorrect combination of parameters that define the inner side of the wheel. **Error correction** Check the parameters of the inner side of the wheel.
280-08A0	**Error message** Dressing roller violates retraction amounts **Cause of error** The cutter width of the dressing roller is greater than the width and retraction amounts AA and AI of the grinding wheel together. Cycle 1018: An edge of the dresser is outside of the retraction amounts AA or AI of the grinding wheel. **Error correction** - Check the retraction amounts of the grinding wheel - Cycle 1018: Also check the center offset
280-08A0	**Error message** FN14_2208 **Cause of error** FN14_2208 **Error correction** FN14_2208
280-08A1	**Error message** FN14_2209 **Cause of error** FN14_2209 **Error correction** FN14_2209

Error number	Description
280-08A2	**Error message** Intermed. dressing not allowed **Cause of error** Intermediate dressing not allowed. Intermediate dressing is not allowed in the present condition of the machine, or no dressing is defined in the running program. **Error correction** Define dressing in the program.
280-08A3	**Error message** FN14_2211 **Cause of error** FN14_2211 **Error correction** FN14_2211
280-08A4	**Error message** FN14_2212 **Cause of error** FN14_2212 **Error correction** FN14_2212
280-08A5	**Error message** FN14_2213 **Cause of error** FN14_2213 **Error correction** FN14_2213
280-08A6	**Error message** FN14_2214 **Cause of error** FN14_2214 **Error correction** FN14_2214
280-08A7	**Error message** FN14_2215 **Cause of error** FN14_2215 **Error correction** FN14_2215

Error number	Description
280-08A8	**Error message** FN14_2216 **Cause of error** FN14_2216 **Error correction** FN14_2216
280-08A9	**Error message** FN14_2217 **Cause of error** FN14_2217 **Error correction** FN14_2217
280-08AA	**Error message** FN14_2218 **Cause of error** FN14_2218 **Error correction** FN14_2218
280-08AB	**Error message** FN14_2219 **Cause of error** FN14_2219 **Error correction** FN14_2219
280-08AC	**Error message** FN14_2220 **Cause of error** FN14_2220 **Error correction** FN14_2220
280-08AC	**Error message** Wheel edge geometry not supported **Cause of error** Dressing cycle and grinding wheel geometry do not match. **Error correction** - Check the wheel geometry - Select a different dressing cycle

Error number	Description
280-08AD	**Error message** FN14_2221 **Cause of error** FN14_2221 **Error correction** FN14_2221
280-08AD	**Error message** Invalid shape of grinding wheel on the outer side **Cause of error** An invalid wheel shape was defined for the outer side of the grinding wheel **Error correction** - Check the wheel shape defined for the outer side - Check the combination of grinding wheel parameters for the outer side
280-08AE	**Error message** FN14_2222 **Cause of error** FN14_2222 **Error correction** FN14_2222
280-08AE	**Error message** Invalid shape of grinding wheel on the inner side **Cause of error** An invalid wheel shape was defined for the inner side of the grinding wheel **Error correction** - Check the wheel shape defined for the inner side - Check the combination of grinding wheel parameters for the inner side
280-08AF	**Error message** FN14_2223 **Cause of error** FN14_2223 **Error correction** FN14_2223

Error number	Description
280-08AF	**Error message** Depth of grinding wheel too large **Cause of error** The depth of the grinding wheel is greater than its radius. This can be caused by dressing. **Error correction** Check the depth of the grinding wheel
280-08B0	**Error message** FN14_2224 **Cause of error** FN14_2224 **Error correction** FN14_2224
280-08B0	**Error message** Dimension of grinding wheel negative **Cause of error** A grinding wheel parameter has become negative. This can be caused by dressing. **Error correction** Check the grinding wheel parameters
280-08B1	**Error message** FN14_2225 **Cause of error** FN14_2225 **Error correction** FN14_2225
280-08B1	**Error message** Minimum value of grinding wheel radius not reached **Cause of error** The current radius of the grinding wheel is smaller than the minimum permissible radius. This can be caused by dressing. **Error correction** Check the grinding wheel parameters
280-08B2	**Error message** FN14_2226 **Cause of error** FN14_2226 **Error correction** FN14_2226

Error number	Description
280-08B3	**Error message** FN14_2227 **Cause of error** FN14_2227 **Error correction** FN14_2227
280-08B4	**Error message** FN14_2228 **Cause of error** FN14_2228 **Error correction** FN14_2228
280-08B5	**Error message** FN14_2229 **Cause of error** FN14_2229 **Error correction** FN14_2229
280-08B6	**Error message** FN14_2230 **Cause of error** FN14_2230 **Error correction** FN14_2230
280-08B7	**Error message** FN14_2231 **Cause of error** FN14_2231 **Error correction** FN14_2231
280-08B8	**Error message** FN14_2232 **Cause of error** FN14_2232 **Error correction** FN14_2232

Error number	Description
280-08B9	**Error message** FN14_2233 **Cause of error** FN14_2233 **Error correction** FN14_2233
280-08BA	**Error message** FN14_2234 **Cause of error** FN14_2234 **Error correction** FN14_2234
280-08BB	**Error message** FN14_2235 **Cause of error** FN14_2235 **Error correction** FN14_2235
280-08BC	**Error message** FN14_2236 **Cause of error** FN14_2236 **Error correction** FN14_2236
280-08BD	**Error message** FN14_2237 **Cause of error** FN14_2237 **Error correction** FN14_2237
280-08BE	**Error message** FN14_2238 **Cause of error** FN14_2238 **Error correction** FN14_2238

Error number	Description
280-08BF	**Error message** FN14_2239 **Cause of error** FN14_2239 **Error correction** FN14_2239
280-08C0	**Error message** FN14_2240 **Cause of error** FN14_2240 **Error correction** FN14_2240
280-08C1	**Error message** FN14_2241 **Cause of error** FN14_2241 **Error correction** FN14_2241
280-08C2	**Error message** FN14_2242 **Cause of error** FN14_2242 **Error correction** FN14_2242
280-08C3	**Error message** FN14_2243 **Cause of error** FN14_2243 **Error correction** FN14_2243
280-08C4	**Error message** FN14_2244 **Cause of error** FN14_2244 **Error correction** FN14_2244

Error number	Description
280-08C5	**Error message** FN14_2245 **Cause of error** FN14_2245 **Error correction** FN14_2245
280-08C6	**Error message** FN14_2246 **Cause of error** FN14_2246 **Error correction** FN14_2246
280-08C7	**Error message** FN14_2247 **Cause of error** FN14_2247 **Error correction** FN14_2247
280-08C8	**Error message** FN14_2248 **Cause of error** FN14_2248 **Error correction** FN14_2248
280-08C9	**Error message** FN14_2249 **Cause of error** FN14_2249 **Error correction** FN14_2249
280-08CA	**Error message** FN14_2250 **Cause of error** FN14_2250 **Error correction** FN14_2250

Error number	Description
280-08CB	**Error message** FN14_2251 **Cause of error** FN14_2251 **Error correction** FN14_2251
280-08CC	**Error message** FN14_2252 **Cause of error** FN14_2252 **Error correction** FN14_2252
280-08CD	**Error message** FN14_2253 **Cause of error** FN14_2253 **Error correction** FN14_2253
280-08CE	**Error message** FN14_2254 **Cause of error** FN14_2254 **Error correction** FN14_2254
280-08CF	**Error message** FN14_2255 **Cause of error** FN14_2255 **Error correction** FN14_2255
280-08D0	**Error message** FN14_2256 **Cause of error** FN14_2256 **Error correction** FN14_2256

Error number	Description
280-08D1	**Error message** FN14_2257 **Cause of error** FN14_2257 **Error correction** FN14_2257
280-08D2	**Error message** FN14_2258 **Cause of error** FN14_2258 **Error correction** FN14_2258
280-08D3	**Error message** FN14_2259 **Cause of error** FN14_2259 **Error correction** FN14_2259
280-08D4	**Error message** FN14_2260 **Cause of error** FN14_2260 **Error correction** FN14_2260
280-08D5	**Error message** FN14_2261 **Cause of error** FN14_2261 **Error correction** FN14_2261
280-08D6	**Error message** FN14_2262 **Cause of error** FN14_2262 **Error correction** FN14_2262

Error number	Description
280-08D7	**Error message** FN14_2263 **Cause of error** FN14_2263 **Error correction** FN14_2263
280-08D8	**Error message** FN14_2264 **Cause of error** FN14_2264 **Error correction** FN14_2264
280-08D9	**Error message** FN14_2265 **Cause of error** FN14_2265 **Error correction** FN14_2265
280-08DA	**Error message** FN14_2266 **Cause of error** FN14_2266 **Error correction** FN14_2266
280-08DB	**Error message** FN14_2267 **Cause of error** FN14_2267 **Error correction** FN14_2267
280-08DC	**Error message** FN14_2268 **Cause of error** FN14_2268 **Error correction** FN14_2268

Error number	Description
280-08DD	**Error message** FN14_2269 **Cause of error** FN14_2269 **Error correction** FN14_2269
280-08DE	**Error message** FN14_2270 **Cause of error** FN14_2270 **Error correction** FN14_2270
280-08DF	**Error message** FN14_2271 **Cause of error** FN14_2271 **Error correction** FN14_2271
280-08E0	**Error message** FN14_2272 **Cause of error** FN14_2272 **Error correction** FN14_2272
280-08E1	**Error message** FN14_2273 **Cause of error** FN14_2273 **Error correction** FN14_2273
280-08E2	**Error message** FN14_2274 **Cause of error** FN14_2274 **Error correction** FN14_2274

Error number	Description
280-08E3	**Error message** FN14_2275 **Cause of error** FN14_2275 **Error correction** FN14_2275
280-08E4	**Error message** FN14_2276 **Cause of error** FN14_2276 **Error correction** FN14_2276
280-08E5	**Error message** FN14_2277 **Cause of error** FN14_2277 **Error correction** FN14_2277
280-08E6	**Error message** FN14_2278 **Cause of error** FN14_2278 **Error correction** FN14_2278
280-08E7	**Error message** FN14_2279 **Cause of error** FN14_2279 **Error correction** FN14_2279
280-08E8	**Error message** FN14_2280 **Cause of error** FN14_2280 **Error correction** FN14_2280

Error number	Description
280-08E9	**Error message** FN14_2281 **Cause of error** FN14_2281 **Error correction** FN14_2281
280-08EA	**Error message** FN14_2282 **Cause of error** FN14_2282 **Error correction** FN14_2282
280-08EB	**Error message** FN14_2283 **Cause of error** FN14_2283 **Error correction** FN14_2283
280-08EC	**Error message** FN14_2284 **Cause of error** FN14_2284 **Error correction** FN14_2284
280-08ED	**Error message** FN14_2285 **Cause of error** FN14_2285 **Error correction** FN14_2285
280-08EE	**Error message** FN14_2286 **Cause of error** FN14_2286 **Error correction** FN14_2286

Error number	Description
280-08EF	**Error message** FN14_2287 **Cause of error** FN14_2287 **Error correction** FN14_2287
280-08F0	**Error message** FN14_2288 **Cause of error** FN14_2288 **Error correction** FN14_2288
280-08F1	**Error message** FN14_2289 **Cause of error** FN14_2289 **Error correction** FN14_2289
280-08F2	**Error message** FN14_2290 **Cause of error** FN14_2290 **Error correction** FN14_2290
280-08F3	**Error message** FN14_2291 **Cause of error** FN14_2291 **Error correction** FN14_2291
280-08F4	**Error message** FN14_2292 **Cause of error** FN14_2292 **Error correction** FN14_2292

Error number	Description
280-08F5	**Error message** FN14_2293 **Cause of error** FN14_2293 **Error correction** FN14_2293
280-08F6	**Error message** FN14_2294 **Cause of error** FN14_2294 **Error correction** FN14_2294
280-08F7	**Error message** FN14_2295 **Cause of error** FN14_2295 **Error correction** FN14_2295
280-08F8	**Error message** FN14_2296 **Cause of error** FN14_2296 **Error correction** FN14_2296
280-08F9	**Error message** FN14_2297 **Cause of error** FN14_2297 **Error correction** FN14_2297
280-08FA	**Error message** FN14_2298 **Cause of error** FN14_2298 **Error correction** FN14_2298

Error number	Description
280-08FB	**Error message** FN14_2299 **Cause of error** FN14_2299 **Error correction** FN14_2299
280-08FC	**Error message** FN22 command faulty **Cause of error** Error from system function: An FN22 command resulted in an error. **Error correction** Cancel the program, correct the parameter of the FN22, and restart.
280-08FD	**Error message** Parameter block does not exist **Cause of error** Parameter block does not exist You tried to activate a nonexistent parameter block for an axis. **Error correction** Select an existing parameter block.
280-08FE	**Error message** Command not allowed **Cause of error** Command not allowed This command is not supported by the control. **Error correction** Do not use the command
280-08FF	**Error message** Command not allowed at this point **Cause of error** Command not allowed at this point A command supported by the control was used in the wrong context or with the wrong condition of the control. This could be, for example, a grinding command within a dressing program. **Error correction** Check/correct the program

Error number	Description
280-0900	**Error message** FN14_2304 **Cause of error** FN14_2304 **Error correction** FN14_2304
280-0901	**Error message** FN14_2305 **Cause of error** FN14_2305 **Error correction** FN14_2305
280-0902	**Error message** FN14_2306 **Cause of error** FN14_2306 **Error correction** FN14_2306
280-0903	**Error message** FN14_2307 **Cause of error** FN14_2307 **Error correction** FN14_2307
280-0904	**Error message** FN14_2308 **Cause of error** FN14_2308 **Error correction** FN14_2308
280-0905	**Error message** FN14_2309 **Cause of error** FN14_2309 **Error correction** FN14_2309

Error number	Description
280-0906	**Error message** FN14_2310 **Cause of error** FN14_2310 **Error correction** FN14_2310
280-0907	**Error message** FN14_2311 **Cause of error** FN14_2311 **Error correction** FN14_2311
280-0908	**Error message** FN14_2312 **Cause of error** FN14_2312 **Error correction** FN14_2312
280-0909	**Error message** FN14_2313 **Cause of error** FN14_2313 **Error correction** FN14_2313
280-090A	**Error message** FN14_2314 **Cause of error** FN14_2314 **Error correction** FN14_2314
280-090B	**Error message** FN14_2315 **Cause of error** FN14_2315 **Error correction** FN14_2315

Error number	Description
280-090C	**Error message** FN14_2316 **Cause of error** FN14_2316 **Error correction** FN14_2316
280-090D	**Error message** FN14_2317 **Cause of error** FN14_2317 **Error correction** FN14_2317
280-090E	**Error message** FN14_2318 **Cause of error** FN14_2318 **Error correction** FN14_2318
280-090F	**Error message** FN14_2319 **Cause of error** FN14_2319 **Error correction** FN14_2319
280-0910	**Error message** FN14_2320 **Cause of error** FN14_2320 **Error correction** FN14_2320
280-0911	**Error message** FN14_2321 **Cause of error** FN14_2321 **Error correction** FN14_2321

Error number	Description
280-0912	**Error message** FN14_2322 **Cause of error** FN14_2322 **Error correction** FN14_2322
280-0913	**Error message** FN14_2323 **Cause of error** FN14_2323 **Error correction** FN14_2323
280-0914	**Error message** FN14_2324 **Cause of error** FN14_2324 **Error correction** FN14_2324
280-0915	**Error message** FN14_2325 **Cause of error** FN14_2325 **Error correction** FN14_2325
280-0916	**Error message** FN14_2326 **Cause of error** FN14_2326 **Error correction** FN14_2326
280-0917	**Error message** Non-circular channel not active **Cause of error** Non-circular channel not active A command was programmed that expects an active non-circular channel (non-circular program). **Error correction** Check/correct the program

Error number	Description
280-0918	**Error message** Non-circ. channel still active **Cause of error** Non-circular channel still active A command was programmed that is not allowed with an active non-circular channel. This could be, for example, an M3, M4 or M5. **Error correction** Check/correct the program
280-0919	**Error message** Command only allowed in non-circular channel **Cause of error** Command only allowed in non-circular channel A command was programmed in a normal program that is allowed only in a contour program (non-circular program). A contour program (non-circular program) was started as a normal program. **Error correction** Check/correct the program Start the non-circular program by means of a cycle and not in full sequence/single block
280-091A	**Error message** Incorrect contour command **Cause of error** Incorrect contour command: The sequence of commands (Cycles 175, 178, 179) is not suited for running a contour program. For example, a contour program is started although none was loaded. **Error correction** Check the sequence of the commands 175, 178 and 179.
280-091B	**Error message** Error in contour machining **Cause of error** Error in contour machining. **Error correction**
280-091C	**Error message** Contour program faulty **Cause of error** Contour program error: A Cycle 176 was programmed at the beginning of a contour program (noncircular contour). There is an error in the combination of possible parameters. **Error correction** Check the parameters of Cycle 176 in the contour program.

Error number	Description
280-092F	**Error message** Tool axis X, Y, Z permitted **Cause of error** Only X, Y, and Z are possible as tool axes **Error correction**
280-0930	**Error message** Starting spindle angle missing **Cause of error** For the imaging of tools, at least the spindle angle of one tooth must be known. **Error correction** - Run TCH PROBE Cycle 624 in order to determine the tooth angles - Enter the spindle angle in the VTC-TOOLS.TAB table
280-0931	**Error message** Max. tilt angle exceeded **Cause of error** The maximum tilting angle of the probe contact was exceeded. **Error correction** Check the setup of the probe contact, and correct it if required. The tilting angle of the probe contact must be within the prescribed limits of both axes.
280-0932	**Error message** Spindle speed not possible **Cause of error** The spindle shaft speed cannot be set for the panorama image. **Error correction** - Please contact your machine tool builder
280-0933	**Error message** Spindle speed not possible **Cause of error** The spindle shaft speed cannot be set for the breakage control. **Error correction** - Please contact your machine tool builder

Error number	Description
280-0934	**Error message** Camera data faulty **Cause of error** Incorrect internal camera data. Caution: Collisions are possible when running the VTC cycles! **Error correction** - Please contact HEIDENHAIN
280-0935	**Error message** Spindle angle unknown **Cause of error** The spindle angles of the tool teeth have not been determined yet. **Error correction** - Run VTC Cycle 624 - Or enter the angles in the TNC:\table\VTC-TOOLS.TAB table
280-0936	**Error message** Option for panorama image is missing **Cause of error** The VTC option for panorama images is not enabled **Error correction** - Consult your contact person at HEIDENHAIN
280-0937	**Error message** Spindle name? **Cause of error** The spindle designation is missing or unknown. **Error correction** - Enter the spindle name in the table PLC:\VTC\VTC.TAB
280-0938	**Error message** Error in VTC.TAB **Cause of error** Entries in the table PLC:\VTC\VTC.TAB are incorrect: - An entry is missing from the feed rate fields - Both FMAX and a numerical value were entered for a feed rate - An incorrect string was entered instead of FMAX **Error correction** Check VTC.TAB and enter the values correctly

Error number	Description
280-093A	**Error message** Excessive number of teeth **Cause of error** The permissible quantity of tool teeth was exceeded. Tools with no more than 32 teeth can be inspected. **Error correction**
280-093B	**Error message** Incorrect VTC API version **Cause of error** The camera cycles do not match the programming interface of the VTC application **Error correction** - Inform your service agency
280-093C	**Error message** Impermissible character in job name **Cause of error** An impermissible character was entered for the job name in the Q string QS620. **Error correction** - Edit the NC program - Check whether any special characters, such as ?, ; , or a blank have been entered
292-0001	**Error message** Look-ahead: internal software error **Cause of error** **Error correction** Inform your service agency.
292-0002	**Error message** ProfilePool: internal software error **Cause of error** **Error correction** Inform your service agency.
292-0003	**Error message** ProfilePool: internal software error **Cause of error** **Error correction** Inform your service agency.

Error number	Description
292-0004	**Error message** ProfilePool: internal software error **Cause of error** **Error correction** Inform your service agency.
292-0005	**Error message** Look-ahead: internal software error **Cause of error** **Error correction** Inform your service agency.
292-0006	**Error message** Thread cutting: Spindle synchronized too late (distance, position controller) **Cause of error** During thread cutting with controlled spindle, the control could not synchronize the spindle soon enough. **Error correction** 1.) Increase the safety clearance 2.) Check the position control loop (kv factor and tolerance window)
292-0007	**Error message** Axes switched while in motion Axes switched while in motion **Cause of error** - Result of emergency stop during movement - Clamping operation was switched while the axis was in motion - Axis was switched off while in motion **Error correction** - If you suspect a PLC error, contact your machine tool builder.
292-0008	**Error message** Incorrect sequence of contour machining **Cause of error** You used an illegal combination of contour machining commands. **Error correction** Correct the NC program

Error number	Description
292-0009	**Error message** %1 **Cause of error** **Error correction**
292-000A	**Error message** Parameters for contour machining not allowed **Cause of error** Control commands for non cylindrical machining are not supported (FN22 FNR720). **Error correction** Inform your service agency
292-000B	**Error message** Clamping mode switched too early **Cause of error** - PLC program has reset PP_AxClampModeRequest (W1038) too early, even before the drives were switched back on. **Error correction** - Inform your machine tool builder.
292-000C	**Error message** System error: No feed rate **Cause of error** A movement without feed rate occurred for no apparent reason. **Error correction** Inform your machine tool builder.
292-000D	**Error message** No spindle available for feed per revolution! **Cause of error** No spindle is available for the tool. **Error correction** Correct the kinematic configuration / switch off M14
292-000E	**Error message** New start after immediate stop of noncylindr. contour not allowed **Cause of error** If the noncylindrical contour was stopped with G179 STOPP=1, a new start with G178 is not allowed. **Error correction** Delete the complete noncylindrical contour with G179 STOPP=3 and reload.

Error number	Description
292-000F	**Error message** Noncircular program too long for RAM **Cause of error** The current noncircular program is too long. Not enough free RAM is available to run the program. **Error correction** Correct or reduce the NC program.
292-0010	**Error message** CMO is not in the current kinematic description CMO is not in the current kinematic description **Cause of error** You tried to activate or deactivate a collision monitored object (CMO) for monitoring. The control cannot fine the CMO in the currently selected kinematics. **Error correction** Correct the name of the CMO to be activated or deactivated.
292-0011	**Error message** Tool is not monitored for collisions **Cause of error** The selected tool has the value zero or a negative value as the radius or length. There will be no monitoring for collisions. **Error correction**
292-0012	**Error message** Tool is not allowed for collision monitoring **Cause of error** You have inserted a tool that cannot be monitored for collision by DCM (e.g. a non-cylindrical tool). **Error correction** Please use another tool.
292-0013	**Error message** General system error in the collision monitoring **Cause of error** A general system error has occurred in the collision monitoring. **Error correction** Inform your service agency.

Error number	Description
292-0014	**Error message** DCM: %1 **Cause of error** Dynamic collision monitoring (DCM) stopped the program in order to avoid a collision. **Error correction**
292-0015	**Error message** Spindle turning too fast **Cause of error** The axis cannot follow because the spindle is turning too fast for the programmed thread. **Error correction** Decrease the spindle speed.
292-0016	**Error message** DCM inactive: Axis has not yet been homed **Cause of error** DCM is inactive because one or more axes have no reference. **Error correction** Do a reference run in the axes
292-0017	**Error message** DCM inactive: RTC (Real Time Coupling) activated **Cause of error** DCM is inactive because for one or more axes the real-time coupling function is active (RTC). **Error correction** Cancel the real-time couplings (RTC)
292-0019	**Error message** Axes have not been homed, although DCM is active **Cause of error** Axes were not yet moved over the reference marks and DCM is switched on. The NC program run was aborted. **Error correction** - Deactivate DCM if non-referenced axes are used

Error number	Description
292-001A	**Error message** Lagged axes with active DCM **Cause of error** In the current NC program, an axis is moved in the following error mode while the DCM collision monitoring is active. The collision monitoring of axes with following error is not supported. The NC program run was aborted. **Error correction** - Edit the NC program, use FUNCTION DCM to deactivate DCM, or - Use the soft key to switch DCM off
292-001B	**Error message** Thread cutting: Spindle synchronization aborted! **Cause of error** During tapping with servo-controlled spindle, the movement was interrupted during spindle synchronization. The tool axis did not come to a stop until after reaching the safety clearance, although the spindle is still turning. **Error correction** - Check the tool and workpiece for possible damage. - Generate the service file and inform your service agency
292-001D	**Error message** DCM inactive: Facing-slide kinematics are activated **Cause of error** DCM collision monitoring is inactive because a facing slide kinematic model was activated. **Error correction** - Deactivate the facing slide kinematics again.
292-001E	**Error message** Facing slide activated with active DCM **Cause of error** The facing slide was activated and collision monitoring (DCM) is switched on. The NC program run was aborted. **Error correction** - Deactivate DCM, if a facing slide is used.

Error number	Description
292-001F	**Error message** Deactivate handwheel for axis %1 Deactivate handwheel for axis %3 **Cause of error** The control waits for automatic clamping of this axis. The activated handwheel is preventing this clamping. **Error correction** Disable the handwheel for this axis
292-0020	**Error message** Configuration of CfgDCM/manualModeDistance is faulty **Cause of error** The value configured for CfgDCM/manualModeDistance is less than the minimum permissible value. **Error correction** Adapt the machine configuration
292-0021	**Error message** Machine configuration is incorrect **Cause of error** The configuration of the parameter profileType or profileTypeHi is faulty. If "advancedTrapezoidal" is to be configured, then this setting must be set in both the profileType parameter as well as in profileTypeHi. The settings currently do not match. **Error correction** Modify the configuration: Configure the value "advancedTrapezoidal" in profileType and in profileTypeHi
292-0022	**Error message** Look-ahead: internal software error. Code %1 **Cause of error** The control has detected an internal software error dealing with the motion control. **Error correction** Inform your service agency
293-0001	**Error message** Internal error in Look-Ahead Chain Module **Cause of error** **Error correction** Inform your service agency.

Error number	Description
293-0002	**Error message** Illegal filter type 2 **Cause of error** The second nominal value filter must not be of the cutter-location type. **Error correction** Adjust the configuration.
293-0003	**Error message** Axis %1 with 2 filters **Cause of error** Two filters are activated in the parameter set of one axis. **Error correction** Adjust the configuration.
293-0004	**Error message** Incorrect filter form in axis %1 **Cause of error** Only position filters with HSC form are allowed. **Error correction** Adjust the configuration
293-0005	**Error message** No cutter-location filter active **Cause of error** The tolerance for rotary axes with M128 was - configured or programmed, although no nominal-value filter of the cutter-location type is active. - configured or programmed, although none of the loaded axis parameters works with the 1st filter. **Error correction** Adjust the configuration or edit the program.
293-0006	**Error message** Tolerance for rotary axes inactive **Cause of error** The tolerance for rotary axes with M128 was deactivated because you switched to a set of axis parameters without cutter-location filter. Therefore the normal path tolerance is being again used for all axes. The tolerance for rotary axes with M128 stays active until the cutter-location filter is used again. **Error correction** A correction is not essential. The warning can be suppressed by deactivating the tolerance for rotary axes before the axis parameter switchover.

Error number	Description
293-0007	**Error message** Axis parameters for logical axis %1 were lost **Cause of error** Internal error The most recently programmed axis parameters of the axis named were lost in an abnormal program cancellation. Therefore the configured parameters will be used. **Error correction** The control continues operation without correction. If you still know the lost axis parameters, you can program them again.
293-0008	**Error message** Parameters from another axis were programmed %1 **Cause of error** The axis for which the axis parameters were programmed do not belong to the channel. **Error correction** Edit the program
293-0009	**Error message** Parameters from another axis were configured %1 **Cause of error** A logical axis is assigned to another, physical axis. The parameter of this physical axis cannot be overwritten by the values of the logical axis. **Error correction** Change the configuration
293-000A	**Error message** Missing axis parameters in CfgAxis of %1 **Cause of error** The configuration of an axis has no entries for the axis parameters used here. **Error correction** Change the configuration
293-000B	**Error message** Two cutter-location filters **Cause of error** Both position filters are of the cutter-location type. **Error correction** Change the configuration.

Error number	Description
293-000C	**Error message** Position filter is missing **Cause of error** With a position filter of the cutter-location type, the second filter must be of the position type. **Error correction** Change the configuration.
293-000D	**Error message** Two position filters in axis %1 **Cause of error** Two position filters of the position type are configured for the same axis. **Error correction** Change the configuration.
293-000E	**Error message** No position filter in axis %1 **Cause of error** For axes with cutter-location filters, the second filters must also be configured. **Error correction** Change the configuration.
293-000F	**Error message** The option for axis-specific jerk has not been enabled **Cause of error** Axis-specific jerk values were activated in the machine configuration, but the option was not yet enabled. Now the jerk that is valid in the channel was activated for the axis. **Error correction** - Delete the parameters MP_axPathJerk and MP_axPathJerkHi for axis-specific jerk from the machine configuration or enable the software option.
293-0010	**Error message** The option for the feed rate filter has not been enabled **Cause of error** A time constant for the feed rate filter was configured in the machine configuration, but the option was not yet enabled. The feed rate filter was deactivated by the NC software. **Error correction** - Delete the parameter MP_filterFeedTime for the time constant of the feed rate filter from the machine configuration or enable the software option.

Error number	Description
293-0011	**Error message** In Program Run it is not allowed to switch the filter on or off **Cause of error** A filter can be switched on or off by changing the filter form, filter order or the frequency. Filter switched on: Form is not "Off" and the order is greater than 1. Filter switched off: Form is "Off" and the order is less than 1. Only for HSC filters: Filter is switched off if frequency = 0. The programmed parameters for the filter were not adopted. **Error correction** - Edit the program or the cycle.
293-0012	**Error message** Faulty configuration of the nominal position value filters **Cause of error** The following filter parameters are no longer supported and must no longer be configured: - CfgFilter/typeFilter1, typeFilter2, orderFilter1, orderFilter2, - CfgPositionFilter/filter1Shape, filter2Shape, filter1LimitFreq, filter2LimitFreq **Error correction** The old parameters of the position nominal value filter are automatically deleted by a correctly run configuration update. The following conditions must be met in order to conduct the update: - The config object CfgFilter must completely match the old level (with no new parameters). - Either only new parameters or only old parameters are permitted for each CfgPositionFilter config object in the system.
293-0013	**Error message** Name of parameter set (%1) for axis (%2) already assigned **Cause of error** Two or more axes are using the same parameter set. The names of parameter sets must be unique for each axis. **Error correction** - Assign a unique name for each of the parameter sets.

Error number	Description
293-0014	**Error message** Option for limiting the radial acceleration not enabled **Cause of error** A maximum value for the radial acceleration was configured in the machine configuration, but the option was not yet enabled. The limit to the radial acceleration was deactivated by the NC software. **Error correction** - Delete the maxTransAcc and maxTransAccHi parameters for limiting the radial acceleration from the machine configuration or enable the software option.
293-0016	**Error message** Filter settings for virtual axis %1 not allowed **Cause of error** An individual filter setting was configured for a virtual axis. That is not allowed for virtual axes. The individual filter setting was deactivated by the NC. An axis is virtual if CfgAxis/axisMode has the value "Virtual". **Error correction** - Configuration: Delete the config object CfgPositionFilter for this axis and use CfgKeySynonym to connect it with the real axis - FN17: Use FN17 to delete filter settings from the NC program - Cycle 32: Delete HSC mode from Cycle 32
293-0017	**Error message** ADP must be active during use of DCM tolerances **Cause of error** The additional tolerances CfgDCM/maxLinearTolerance and CfgDCM/maxAngleTolerance are configured and ADP is not active. This is not allowed. **Error correction** Switch-on ADP by configuring CfgHardware/setupADP = Premium. Comply with the information on ADP in the Technical Manual.

Error number	Description
293-0018	**Error message** Excessive contour error (%1 mm) in thread (%2 mm permitted) **Cause of error** The adjusted tolerance for successive threads was exceeded **Error correction** - Edit the NC program If possible, reduce the spindle speed. - If a larger deviation is acceptable, increase the tolerance for threads. - Inform your service agency.
293-0019	**Error message** End of successive threads not reached **Cause of error** The programmed end point of the successive thread was not reached. **Error correction** - Check the NC program and edit it if necessary - Reduce the spindle speed, if possible
293-001A	**Error message** Invalid number programmed for FN17/18 No%1 **Cause of error** - Incorrect number of FN17/18 programmed during access to look-ahead parameters. - The is no look-ahead parameter under this number. **Error correction** - Correct the corresponding FN17 or FN18 command.
293-001B	**Error message** Invalid axis programmed in FN17/18 IDX%1 **Cause of error** - Incorrect axis/spindle was programmed through FN17 or FN 18 during access to look-ahead parameters. - There is no axis or spindle with this axis index. **Error correction** - Correct the corresponding FN17 or FN18 command.

Error number	Description
293-001C	**Error message** Vertical movement not allowed in the middle of successive threads **Cause of error** - A movement perpendicular to the threaded axis was programmed in the middle of the thread. - A perpendicular movement is allowed only in the form of lift-off at the end of the thread. **Error correction** - Check the NC program and adapt it if necessary
293-001E	**Error message** Programmed feed rate too small **Cause of error** The programmed feed rate is too small. **Error correction** Edit the NC program.
293-001F	**Error message** The NC program is longer than %1 **Cause of error** The total distance moved in the NC program is greater than permitted. **Error correction** Shorten the NC program. Remove infinite loops.
293-0020	**Error message** Internal error in LookAheadChain module. Code %1 **Cause of error** The control has detected an internal software error dealing with the motion control. **Error correction** Inform your service agency
293-0021	**Error message** Impermissible dynamic calculation of non-registered axis %1 **Cause of error** Look-ahead is supposed to calculate the dynamics of an axis that is not registered. The cause could be a deactivated axis that is used in the kinematics. That is not allowed. **Error correction** - Activate the deactivated axis, check the machine configuration, and correct it if required - Activate another machine kinematic configuration through the NC program - Edit the machine configuration or activate another machine kinematic configuration

Error number	Description
293-0022	**Error message** NC program contains more than %1 blocks **Cause of error** The NC program contains too many blocks. **Error correction** Shorten the NC program. Remove infinite loops.
2A0-0001	**Error message** Ext. in-/output not ready **Cause of error** - The interface is not connected. - The external device is either switched off or not ready. - The transmission cable is defective or incorrect. **Error correction** Check the data transfer line.
2A0-0002	**Error message** Error **Cause of error** This message indicates that there is an error message on the screen now in the background. **Error correction** Switch to the background mode and acknowledge the error message.
2A0-0003	**Error message** Interface already assigned **Cause of error** You attempted to assign an already occupied data interface. **Error correction** End the data transmission and restart it.
2A0-0004	**Error message** Baud rate not possible **Cause of error** The baud rates set at the two data interfaces do not permit simultaneous transmission over both interfaces. **Error correction** Select another baud rate.

Error number	Description
2A0-0005	**Error message** Data transfer erroneous **Cause of error** E During data transfer with BCC the <NAK> signal was received 15 times in succession. A to H Error code of the receiver module with one w/o E of the following causes: - The baud rate settings of the TNC and peripheral device do not match. - The parity bit is erroneous. - Erroneous data frame (e.g.: no stop-bit). - The receiver module of the interface is defective. K During transmission of an error to the TNC the <1> character was not transmitted after the <ESC> character. L After the error sequence <ESC><1> an incorrect error number was received (error numbers 0 to 7 are permitted). M During data transmission with BCC the <NAK> character was transmitted 15 times in succession. N An expected acknowledgment <ACK> or <NAK> was not transmitted after a certain time. **Error correction** Check the data transfer channel.
2A0-0006	**Error message** LSV2: Line interrupted **Cause of error** - DSR signal missing **Error correction** - Check the data transfer line
2A0-0007	**Error message** LSV2: Transmission error **Cause of error** - Character error in the telegram **Error correction** - Check the data transfer line
2A0-0008	**Error message** LSV2: Transmission error **Cause of error** - Checksum error in the received telegram **Error correction** - Check the data transfer line - If the error recurs, inform your service agency

Error number	Description
2A0-0009	**Error message** LSV2: Transmission error **Cause of error** - Checksum error in the transmitted telegram **Error correction** - Check the data transfer line - If the error recurs, inform your service agency
2A0-000A	**Error message** LSV2: Timeout error **Cause of error** - No reaction from distant terminal (T1) **Error correction** - If the error recurs, inform your service agency - Check the LSV2TIME1 entry in OEM.SYS
2A0-000B	**Error message** LSV2: Transmission error **Cause of error** - Distant terminal not ready **Error correction** - Check the communications software of distant terminal
2A0-000C	**Error message** LSV2: Timeout error **Cause of error** - Telegram incomplete, ETX missing (T0) **Error correction** - Check the communications software of distant terminal - If the error recurs, inform your service agency - Check the LSV2TIME0 entry in OEM.SYS
2A0-000D	**Error message** LSV2: Timeout error **Cause of error** - No reaction from distant terminal (T2) **Error correction** - Check the communications software of distant terminal - If the problem recurs, inform your service agency - Check the LSV2TIME2 entry in OEM.SYS

Error number	Description
2A0-000E	**Error message** LSV2: Outgoing transmiss. error **Cause of error** - Internal software error **Error correction** - If the problem recurs, inform your service agency - Check the software version
2A0-0010	**Error message** Illegal file name **Cause of error** Syntax error during file-name input. **Error correction** Use no more than 16 characters for file names.
2A0-0011	**Error message** Key is locked **Cause of error** You pressed a key that is locked at present by the NC software. **Error correction** If necessary, repeat the function at a later time.
2A0-0012	**Error message** Function not permitted **Cause of error** You tried to use a feature that is not enabled on your control by the Feature Content Level (FCL) management. **Error correction** By default, FCL functions are locked after a software update. By entering the code number 65535 in the SIK Menu, you can enable these functions for a certain period of time for test purposes. You can enable FCL functions permanently by purchasing and entering a code number. For more information, contact your machine tool builder or the service agency for the control.
2A0-0013	**Error message** Two rotary positions not allowed **Cause of error** In a pattern or a frame, you tried to define two rotary positions at once. **Error correction** Define only the rotary position of the reference axis or minor axis.

Error number	Description
2A0-0014	**Error message** File name already exists **Cause of error** You tried to assign a name to a new file, although that name already exists. **Error correction** Use another file name.
2A0-0015	**Error message** Function not enabled **Cause of error** You tried to use a software option that is not enabled on your control. **Error correction** Contact your machine tool builder or the service agency for the control to purchase the software option.
2A0-0016	**Error message** Caution: Prepos. height defined! **Cause of error** You have hidden or disabled a point for which a pre-positioning height is defined. In some cases this could lead to a collision. **Error correction** Check if the next points can be approached without danger.
2A0-0017	**Error message** Too little free memory on SYS: **Cause of error** There is not enough memory on the system partition SYS:. Proper function of the control can no longer be ensured. **Error correction** - Reboot the control. - Inform your service agency
2A0-0018	**Error message** Too little free memory on SYS: **Cause of error** There is not enough memory on the system partition SYS:. Proper function of the control can no longer be ensured. **Error correction** - Reboot the control. - Inform your service agency

Error number	Description
2A0-0019	**Error message** Too little free memory on SYS: **Cause of error** There is not enough memory on the system partition SYS:. Proper function of the control can no longer be ensured. **Error correction** - Reboot the control. - Inform your service agency
2A0-001A	**Error message** Too little free memory on SYS: **Cause of error** There is not enough memory on the system partition SYS:. Proper function of the control can no longer be ensured. **Error correction** - Reboot the control. - Inform your service agency
2A0-001B	**Error message** Too little free memory on SYS: **Cause of error** There is not enough memory on the system partition SYS:. Proper function of the control can no longer be ensured. **Error correction** - Reboot the control. - Inform your service agency
2A0-001C	**Error message** Too little free memory on PLC: **Cause of error** There is not enough memory on the PLC: partition. Proper function of the control can no longer be ensured. **Error correction** Inform your machine tool builder.
2A0-001D	**Error message** Too little free memory on PLC: **Cause of error** There is not enough memory on the PLC: partition. Proper function of the control can no longer be ensured. **Error correction** Inform your machine tool builder.

Error number	Description
2A0-001E	**Error message** Too little free memory on PLC: **Cause of error** There is not enough memory on the PLC: partition. Proper function of the control can no longer be ensured. **Error correction** Inform your machine tool builder.
2A0-001F	**Error message** Too little free memory on PLC: **Cause of error** There is not enough memory on the PLC: partition. Proper function of the control can no longer be ensured. **Error correction** Inform your machine tool builder.
2A0-0020	**Error message** Too little free memory on PLC: **Cause of error** There is not enough memory on the PLC: partition. Proper function of the control can no longer be ensured. **Error correction** Inform your machine tool builder.
2A0-0021	**Error message** Too little free memory on TNC: **Cause of error** You saved too many files or excessively large files on the TNC: partition of the control. If you save any more files you will prevent safe operation of the control. **Error correction** - Delete NC programs that are no longer required - Delete backup files of NC programs (*.bak) that are no longer required - Delete service files that are no longer required - If a machining process is now in progress, do not under any circumstance save any more files. It would endanger the success of the operation.

Error number	Description
2A0-0022	**Error message** Too little free memory on TNC: **Cause of error** You saved too many files or excessively large files on the TNC: partition of the control. If you save any more files you will prevent safe operation of the control. **Error correction** - Delete NC programs that are no longer required - Delete backup files of NC programs (*.bak) that are no longer required - Delete service files that are no longer required - If a machining process is now in progress, do not under any circumstance save any more files. It would endanger the success of the operation.
2A0-0023	**Error message** Too little free memory on TNC: **Cause of error** You saved too many files or excessively large files on the TNC: partition of the control. If you save any more files you will prevent safe operation of the control. **Error correction** - Delete NC programs that are no longer required - Delete backup files of NC programs (*.bak) that are no longer required - Delete service files that are no longer required - If a machining process is now in progress, do not under any circumstance save any more files. It would endanger the success of the operation.
2A0-0024	**Error message** Too little free memory on TNC: **Cause of error** You saved too many files or excessively large files on the TNC: partition of the control. If you save any more files you will prevent safe operation of the control. **Error correction** - Delete NC programs that are no longer required - Delete backup files of NC programs (*.bak) that are no longer required - Delete service files that are no longer required - If a machining process is now in progress, do not under any circumstance save any more files. It would endanger the success of the operation.

Error number	Description
2A0-0025	**Error message** Too little free memory on TNC: **Cause of error** You saved too many files or excessively large files on the TNC: partition of the control. If you save any more files you will prevent safe operation of the control. **Error correction** - Delete NC programs that are no longer required - Delete backup files of NC programs (*.bak) that are no longer required - Delete service files that are no longer required - If a machining process is now in progress, do not under any circumstance save any more files. It would endanger the success of the operation.
2A0-0026	**Error message** Not enough free RAM. **Cause of error** There is little free RAM available. If you use any more RAM you will prevent safe operation of the control. **Error correction** - Deactivate memory-intensive applications such as the editor graphics or the Test Run mode. - Reboot the control.
2A0-0027	**Error message** Not enough free RAM. **Cause of error** There is little free RAM available. If you use any more RAM you will prevent safe operation of the control. **Error correction** - Deactivate memory-intensive applications such as the editor graphics or the Test Run mode. - Reboot the control.
2A0-0028	**Error message** Not enough free RAM. **Cause of error** There is little free RAM available. If you use any more RAM you will prevent safe operation of the control. **Error correction** - Deactivate memory-intensive applications such as the editor graphics or the Test Run mode. - Reboot the control.

Error number	Description
2A0-0029	**Error message** Not enough free RAM. **Cause of error** There is little free RAM available. If you use any more RAM you will prevent safe operation of the control. **Error correction** - Deactivate memory-intensive applications such as the editor graphics or the Test Run mode. - Reboot the control.
2A0-002A	**Error message** Not enough free RAM. **Cause of error** There is little free RAM available. If you use any more RAM you will prevent safe operation of the control. **Error correction** - Deactivate memory-intensive applications such as the editor graphics or the Test Run mode. - Reboot the control.
2A0-002B	**Error message** Cannot switch modes of operation **Cause of error** You started the pattern generator or the contour programming while in a machining form and from there tried to switch to another operating mode. **Error correction** Close the pattern generator or contour programming and then close the form input mode (saving or discarding the data) before you switch to another operating mode.
2A0-002C	**Error message** Auto. keyboard lock was opened **Cause of error** The TNC locks keys during a status change, but the lock was opened because the status change was not completed within 15 seconds. **Error correction** Wait until the status change is complete.

Error number	Description
2A0-002D	**Error message** Tree view buildup canceled **Cause of error** After selecting a long smarT.NC program, you canceled the buildup of the tree view. The TNC therefore cannot show the complete tree structure of the program within smarT.NC, nor can you test or run the program. **Error correction** Select the program again and this time wait until the TNC has finished building the tree view.
2A0-002E	**Error message** PGM being edited in parallel **Cause of error** You tried to edit form data that are being edited in the Programming and Editing mode. **Error correction** Cancel your editing in the Programming and Editing mode and make the desired changes in the smarT.NC mode.
2A0-002F	**Error message** Program header already exists **Cause of error** You tried to insert the UNIT 700 (program header), although it already exists. **Error correction** Make changes in the existing program header.
2A0-0030	**Error message** Clipboard is empty! **Cause of error** You tried to insert a block from the clipboard, but the clipboard is empty. **Error correction** First fill the clipboard with the COPY BLOCK or CUT BLOCK.
2A0-0031	**Error message** System memory overflow **Cause of error** This error occurs when the TNC does not have enough buffer memory for calculations, e.g. for generating complex FK graphics while machining a complex part. **Error correction** Acknowledge the error message by pressing CE and repeat the function.

Error number	Description
2A0-0032	**Error message** Function not permitted! **Cause of error** You tried to use a workpiece blank definition from an .hu program, but you did not select the current contour program from an .hu program. **Error correction** Use the function only when you have started the contour program from a UNIT program.
2A0-0033	**Error message** File does not exist **Cause of error** You tried to use the "Recent Files" function to open a file that was moved or no longer exists. **Error correction** Select a different file or open a new file.
2A0-0034	**Error message** File format has changed **Cause of error** This error message will be displayed upon opening a binary file (*.H,*.T...) if the binary format has changed since the previous output version. **Error correction** Delete the file.
2A0-0035	**Error message** smarT.NC: Programming smarT.NC: Programming **Cause of error** **Error correction**
2A0-0036	**Error message** smarT.NC: Defining contours smarT.NC: Contour def. **Cause of error** **Error correction**

Error number	Description
2A0-0037	**Error message** smarT.NC: Defining positions smarT.NC: Position def. **Cause of error** **Error correction**
2A0-0038	**Error message** Application %1 cannot be started **Cause of error** - Due to a lack of system resources (e.g. during memory-intensive HSC machining), a part of the system software cannot be loaded. - The TNC cannot interpret the format of a DXF file. - The DXF file is destroyed. **Error correction** - Call the desired function again at a later time. - Ensure that the DXF file is available in R12 (ASCII) format. - If required, recreate the DXF file.
2A0-0039	**Error message** %1 **Cause of error** **Error correction**
2A0-003A	**Error message** Programming graphics impossible **Cause of error** Programming graphics generation had to be terminated due to an internal error. **Error correction** Select the NC program again and restart generation of a new programming graphic (RESET+START soft key)
2A0-003B	**Error message** smarT.NC: Select DXF elements smarT.NC: Select elements **Cause of error** **Error correction**

Error number	Description
2A0-003C	**Error message** Erroneous DXF file Use of DXF_CONVERTER not possible **Cause of error** You tried to open a DXF file that cannot be edited by the TNC. **Error correction** - Check whether the DXF file is available in ASCII format. - Have the DXF file read out in AutoCAD R12 (AC1009€) format. If the problem recurs, try to make the DXF file with another CAD system. - If necessary, inform your service agency.
2A0-003E	**Error message** Program-run graphics impossible! **Cause of error** The TNC is being so heavily utilized to machine the current workpiece that there is no more capacity available for the program-run graphics. **Error correction** No corrective action possible.
2A0-003F	**Error message** Global PGM settings deactivated **Cause of error** In the smarT.NC operating mode you selected the Program Run submode although global program settings were active. **Error correction** The TNC automatically deactivates all active global program settings. Reactivate the settings, if necessary, when you continue working in the Program Run, Single Block or Program Run, Full Sequence operating mode.
2A0-0040	**Error message** smarT.NC: Program run smarT.NC: Program run **Cause of error** **Error correction**
2A0-0041	**Error message** To retract: NC start **Cause of error** **Error correction**

Error number	Description
2A0-0042	**Error message** Selected block not addressed **Cause of error** After an interruption of the program run the TNC can no longer resume the program run from the cursor's present location. **Error correction** Press GOTO and enter a block number to select the desired location for returning to the program, or select the mid-program startup function.
2A0-0043	**Error message** Test graphic not possible **Cause of error** At present, the system is preventing use of the test graphic. **Error correction** Use the test graphic at a later time.
2A0-0044	**Error message** smarT.NC: Testing smarT.NC: Testing **Cause of error** **Error correction**
2A0-0045	**Error message** Tool file? **Cause of error** There are several tool tables in the NC memory and no table is activated in the Test Run operating mode. **Error correction** Activate the tool table in the Test Run operating mode (status "S").
2A0-0047	**Error message** GOTO table line **Cause of error** **Error correction**

Error number	Description
2A0-0048	**Error message** Locked axis was programmed **Cause of error** - You programmed a locked axis in a part program block. - A traverse was calculated for a locked axis (e.g. due to an active rotation). - A programmed axis is a freely traversing rotary axis. **Error correction** - If necessary, activate the axis. - Delete the axis from the part program block.
2A0-0049	**Error message** Spindle must be turning Spindle must be turning **Cause of error** You called a fixed cycle without first switching on the spindle. **Error correction** Edit the part program.
2A0-004A	**Error message** Tool axis is missing **Cause of error** You called a fixed cycle without first activating a tool. **Error correction** Edit the part program.
2A0-004B	**Error message** Tool radius too small Tool radius too small **Cause of error** - The tool radius is too small for the selected operation. - Cycle 3 "Slot": You defined a width greater than four times the tool radius. - Cycle 240: You entered a centering diameter greater than the tool diameter. - Cycle 210 "Slot" or Cycle 211 "Circular Slot": The slot width is six times greater than the tool radius. **Error correction** - Use a tool with a larger radius. - Cycle 3 "Slot": Define the slot width to be greater than the tool diameter and smaller than four times the tool radius. - Cycle 240: Use a larger tool. - Cycle 210 "Slot" or Cycle 211 "Circular Slot": Define the slot width to be greater than the tool diameter and smaller than six times the tool radius.

Error number	Description
2A0-004C	**Error message** Tool radius too large Tool radius too large **Cause of error** - Contour milling: The radius of an arc block at an inside corner is smaller than the tool radius. - Thread milling: The thread core diameter is smaller than the tool diameter. - Slot milling: The slot width for roughing is smaller than the tool diameter. - Cycle 251, rectangular pocket: The rounding radius Q220 is smaller than the tool radius. - Cycle 214: The given workpiece-blank diameter is smaller than the tool diameter **Error correction** - Use a smaller tool - Slot milling: If necessary, use a smaller oversize (Q368) - Cycle 214: Use a smaller tool; correct the workpiece-blank diameter
2A0-004D	**Error message** Range exceeded Range exceeded **Cause of error** During digitizing the stylus went out of the defined digitizing range. **Error correction** Check the data in the Range cycle, especially the entry for the touch probe axis.
2A0-004E	**Error message** Start position incorrect Start position incorrect **Cause of error** Digitizing with contour lines: Incorrect starting position selected. **Error correction** Check the axes defined in the Contour Lines cycle.

Error number	Description
2A0-004F	**Error message** Rotation not permitted **Cause of error** - Rotation not permitted during digitizing. - Rotation not permitted during automatic measuring (measuring cycles 400 to 418) together with 3-D rotation. - 3-D rotation not permitted together with Cycle 247. **Error correction** - Delete the Rotation cycle. - Reset the rotation (manual mode). - Reset 3-D rotation.
2A0-0050	**Error message** Scaling factor not permitted **Cause of error** You programmed a scaling factor before the TCH PROBE 0 cycle (ISO: G55) or before the digitizing cycles. **Error correction** Delete the Scaling Factor or the Axis-Specific Scaling
2A0-0051	**Error message** Mirroring not permitted **Cause of error** You programmed a mirror image before the TCH PROBE 0 cycle (ISO: G55) or before the digitizing cycles. **Error correction** Delete the Mirror Image cycle.
2A0-0052	**Error message** Datum shift not permitted **Cause of error** Digitizing with contour lines: Datum shift is active. **Error correction** Delete the datum shift.
2A0-0053	**Error message** Feed rate is missing Feed rate is missing **Cause of error** You did not program a feed rate. **Error correction** Edit the NC program. FMAX is effective only for the block in which it is programmed.

Error number	Description
2A0-0055	**Error message** Contradictory signs in cycle **Cause of error** The algebraic signs of the setup clearance, total hole depth and plunging depth do not match. **Error correction** Enter identical signs.
2A0-0056	**Error message** Entered angle not permitted Entered angle not permitted Entered angle not permitted **Cause of error** - The solid angles programmed in Cycle 19 Tilt Working Plane (DIN/ISO: G80) cannot be realized with the current attachment (e.g. universal head where only one hemisphere is accessible). - Run probing cycle only with paraxial angular position. - The point angle (T-ANGLE) defined for the active tool is 180°. **Error correction** - Edit the solid angle entered. - Run probing cycle only with paraxial angular position. - Use angular values greater than 0 and less than 180°.
2A0-0057	**Error message** Touch point inaccessible **Cause of error** In the TCH PROBE 0 (ISO: G55) cycle or during use of the manual probe cycles no touch point was reached within the traverse defined in machine parameter MP6130. **Error correction** - Pre-position the touch probe to the workpiece. - Increase the value in MP6130.
2A0-0058	**Error message** Too many points Too many points **Cause of error** Automatic establishment of points for the digitizing range in the Positioning with Manual Data Input operating mode: Number of stored points (max. 893) exceeded. **Error correction** Re-record digitizing range after increasing the point spacing.

Error number	Description
2A0-005A	**Error message** CYCL DEF incomplete CYCL DEF incomplete **Cause of error** - You deleted part of a cycle. - You have inserted other part program blocks within a cycle. **Error correction** - Redefine the complete cycle again - Delete part program blocks programmed within a cycle.
2A0-005B	**Error message** Height axis not permitted here Plane wrongly defined **Cause of error** While defining the Contour Lines cycle (TCH PROBE 7) you programmed a height axis in the starting point. **Error correction** Edit the part program.
2A0-005C	**Error message** Wrong axis programmed Wrong axis programmed **Cause of error** - An incorrect axis is programmed in the highlighted block. - Touch Probe Cycle 403: You programmed in incorrect compensation axis (Q312). **Error correction** - Check whether you have programmed an axis twice. - Touch Probe Cycle 403: In parameter Q312, select only compensation axes that are present in the kinematic description.
2A0-005D	**Error message** Wrong rpm **Cause of error** You entered an invalid spindle speed. **Error correction** Enter the correct speed, refer to the machine manual.

Error number	Description
2A0-005E	**Error message** Radius comp. undefined **Cause of error** - In the definition of a contour, a contour pocket or a contour train you neglected to program radius compensation. - You have called a machining cycle with the tool radius 0. **Error correction** - Set a tool radius compensation in the contour subprogram to define whether the contour is for a pocket or island. - Define a tool radius other than 0.
2A0-005F	**Error message** Rounding-off undefined **Cause of error** You programmed in sequence a positioning block without radius compensation, a rounding arc (RND, ISO: G25), and a circle block with radius compensation. **Error correction** Edit the part program.
2A0-0060	**Error message** Rounding radius too large Rounding radius too large **Cause of error** - In the definition of a contour, a contour pocket or a contour train, you programmed a rounding arc (RND, ISO: G25) with so large a radius that it does not fit between the adjoining elements. - In a fixed cycle (rectangular pocket/rectangular stud), you defined a rounding arc that cannot be inserted. **Error correction** - Define a smaller rounding radius in the contour subprogram - Check the cycle definition and correct the input values
2A0-0061	**Error message** Program start undefined **Cause of error** The TNC cannot exactly calculate the geometry from the present position (e.g., the programmed coordinates of the first positioning block are the same as the compensated actual position). **Error correction** - Restart the part program. - Use mid-program startup to return to the point of interruption.

Error number	Description
2A0-0062	**Error message** Excessive subprogramming **Cause of error** In a Contour Pocket cycle or a Contour Train cycle you called more than 6 programs (PGM CALL, ISO: %..). A program call can also be: - Cycle 12 (PGM CALL, ISO: G39) - Calling an OEM cycle **Error correction** Edit the part program.
2A0-0063	**Error message** Angle reference missing Angle reference missing Angle reference missing **Cause of error** In an LP/CP block (ISO: G10, G11, G12, G13) no polar angle or incremental polar angle is defined, i.e.: - The distance between the last programmed position and the pole is less than or equal to 0.1 µm. - No rotation is programmed between pole assumption and an LP/CP block. **Error correction** - Program the absolute polar angle. - Check the position of the pole. - If necessary, reset the rotation.
2A0-0064	**Error message** No fixed cycle defined No fixed cycle defined **Cause of error** There is no fixed cycle defined before Cycle 220/221 (circular/linear point pattern). **Error correction** Define a fixed cycle before Cycle 220/221.
2A0-0065	**Error message** Insufficient slot width Insufficient slot width **Cause of error** The width defined in the slot cycle cannot be machined with the active tool. **Error correction** Use a smaller tool.

Error number	Description
2A0-0066	**Error message** Pocket too small Pocket too small **Cause of error** The side lengths defined in the Pocket Milling cycle are too small. **Error correction** Use a smaller tool.
2A0-0067	**Error message** Q202 not defined Q202 not defined **Cause of error** There is no plunging depth (Q202) defined in the fixed cycles 200 to 215. **Error correction** Enter a plunging depth in the fixed cycle.
2A0-0068	**Error message** Q205 not defined Q205 not defined **Cause of error** In the Universal Drilling cycle, you have not defined the minimum plunging depth. **Error correction** Enter a minimum plunging depth in the fixed cycle.
2A0-0069	**Error message** Q218 must be greater than Q219 Q218 must be greater than Q219 **Cause of error** Pocket milling cycle: Q218 must be greater than Q219. **Error correction** Correct the values in the fixed cycle.
2A0-006A	**Error message** CYCL 210 not permitted Fixed cycle not allowed **Cause of error** Fixed cycle cannot be run in the CIRCULAR PATTERN or LINEAR PATTERN cycle. **Error correction** Use another fixed cycle.

Error number	Description
2A0-006B	**Error message** CYCL 211 not permitted CYCL 211 not permitted **Cause of error** Cycle 211 cannot be run in the CIRCULAR PATTERN or LINEAR PATTERN cycle. **Error correction** Use another fixed cycle.
2A0-006C	**Error message** Q220 too large Q220 too large **Cause of error** Pocket finishing or stud finishing cycle: Rounding radius Q220 is too large. **Error correction** Correct the rounding radius in the fixed cycle.
2A0-006D	**Error message** Q222 must be greater than Q223 Q222 must be greater than Q223 **Cause of error** Stud finishing cycle: Workpiece blank diameter Q222 must be greater than the finished part diameter Q223. **Error correction** Correct the workpiece blank diameter in the fixed cycle.
2A0-006E	**Error message** Q244 must be greater than 0 Q244 must be greater than 0 **Cause of error** Circular Pattern cycle: You entered a pitch circle diameter of zero. **Error correction** Correct the pitch circle diameter in the cycle.
2A0-006F	**Error message** Q245 must not equal Q246 Q245 must not equal Q246 **Cause of error** Circular Pattern cycle: Enter a stopping angle equal to the starting angle. **Error correction** Correct the starting or stopping angle in the cycle.

Error number	Description
2A0-0070	**Error message** Angle range must be under 360° Angle range must be under 360° **Cause of error** Circular Pattern cycle: You entered an angle range greater than 360°. **Error correction** Correct the starting or stopping angle in the cycle.
2A0-0071	**Error message** Q223 must be greater than Q222 Q223 must be greater than Q222 **Cause of error** In the Circular Pocket Finishing cycle, you entered a finished-part diameter (Q223) smaller than the workpiece-blank diameter (Q222). **Error correction** Edit Q222 in the cycle definition.
2A0-0072	**Error message** Q214: 0 not permitted Q214: 0 not permitted **Cause of error** In the definition of Cycle 204 you have entered the disengaging direction 0. **Error correction** In Q214, enter a value from 1 to 4.
2A0-0089	**Error message** No measuring axis defined No measuring axis defined **Cause of error** You failed to define the measuring axis in one of the measuring cycles 400, 402, 420, 425, 426 or 427. **Error correction** Check Q272 in the corresponding cycle. Permissible input values: 1 or 2; for Cycle 427: 1, 2 or 3.
2A0-008A	**Error message** Tool breakage tolerance exceeded Tool breakage tolerance exceeded **Cause of error** During workpiece inspection using a measuring cycle, the tool breakage tolerance RBREAK given in the tool table was exceeded. **Error correction** Check whether the tool is damaged.

Error number	Description
2A0-008B	**Error message** Enter Q247 unequal 0 Enter Q247 unequal 0 **Cause of error** In a measuring cycle you entered in parameter Q247 an angular step of 0. **Error correction** Enter an angular step (Q247) other than 0.
2A0-008C	**Error message** Enter Q247 greater than 5 Enter Q247 greater than 5 **Cause of error** In a measuring cycle, you entered in parameter Q247 an angular step smaller than 5 degrees. **Error correction** To ensure sufficient measuring accuracy, enter an angular step (Q247) greater than 5 degrees.
2A0-008D	**Error message** Datum table? **Cause of error** A datum table is required to machine a part program. Either there is no table in the control's NC memory, or several tables have be saved and none activated. **Error correction** Activate the datum table in the Program Run, Full Sequence mode (status M).
2A0-008E	**Error message** Enter direction Q351 unequal 0 Enter direction Q351 unequal 0 **Cause of error** In a fixed cycle you did not define the cutting direction (climb or up-cut). **Error correction** Define the cutting direction as climb milling (= 1) or up-cut milling (= -1).
2A0-008F	**Error message** Thread depth too large Thread depth too large **Cause of error** The programmed thread depth plus 1/3 of the pitch is greater than the drilling or sinking depth. **Error correction** Program the total hole depth to be at least 1/3 of a thread pitch smaller that the total hole depth.

Error number	Description
2A0-0090	**Error message** Missing calibration data Missing calibration data **Cause of error** You have attempted to perform a measurement with Cycle 440 without first performing a calibration. **Error correction** Repeat Cycle 440, but with Q363 = 0 (calibrate).
2A0-0093	**Error message** ORIENTATION not permitted **Cause of error** - Your machine does not offer spindle orientation - Spindle orientation not possible **Error correction** - Refer to your machine manual! - Check machine parameter 7442 and enter the numerical designation of the M function or -1 for spindle orientation by the NC. Refer to your machine manual!
2A0-0096	**Error message** Check the depth sign **Cause of error** **Error correction**
2A0-00A4	**Error message** Switchover of Q399 not allowed **Cause of error** You tried to switch on the touch probe cycle 441 to switch on the angle tracking, although this function is deactivated by machine parameter 6165. **Error correction** Set the machine parameter 6165 = 1 (MOD function, code number 123) and then recalibrate the touch probe.
2A0-00A5	**Error message** Tool not defined Tool not defined Tool not defined Tool not defined Tool not defined **Cause of error** You have called a tool that is not defined in the tool table. **Error correction**

Error number	Description
2A0-00A6	**Error message** Tool number not allowed Tool number not allowed **Cause of error** In a TOOL CALL or TOOL DEF block you tried to define a tool number although it is prohibited by machine parameter. **Error correction** - Use the tool name. - Adapt machine parameter 7483. If required, contact your machine manufacturer.
2A0-00A7	**Error message** Tool name not allowed Tool name not allowed **Cause of error** In a TOOL CALL or TOOL DEF block you tried to define a tool name although it is prohibited by machine parameter. **Error correction** - Use the tool number. - If required, contact your machine tool builder
2A0-00AD	**Error message** Handwheel inactive **Cause of error** **Error correction**
2A0-00AE	**Error message** HR not allowed **Cause of error** **Error correction**
2A0-00AF	**Error message** Wrong operating mode for handwheel **Cause of error** **Error correction**
2A0-00B0	**Error message** Manual **Cause of error** **Error correction**

Error number	Description
2A0-00B1	**Error message** Handwheel **Cause of error** **Error correction**
2A0-00B2	**Error message** Manual input **Cause of error** **Error correction**
2A0-00B3	**Error message** Single Block **Cause of error** **Error correction**
2A0-00B4	**Error message** Full Sequence **Cause of error** **Error correction**
2A0-00B5	**Error message** Edit table **Cause of error** **Error correction**
2A0-00B6	**Error message** T%s: Remaining tool life too short **Cause of error** The remaining tool life (TIME2 column in the tool table) of the tool indicated in the error text has been exceeded. **Error correction** - Use a new tool. - Correct the current tool life (CUR.TIME column in the tool table). A complete list of the tools whose tool life has expired is contained in the TOOLLIST.ERR file stored in the TNC:\ directory.

Error number	Description
2A0-00B8	**Error message** FN 14: error code %-3u FN 14: Error code %s **Cause of error** Forced error through function FN14 (ISO: D14). With this function the TNC calls the preprogrammed messages of the machine manufacturer (e.g. from an OEM cycle). If during a program run or test run the TNC comes to a block with FN14 (D14), it stops operation and displays a message. You must then restart the program. **Error correction** Refer to the User's Manual for a description of the error. Correct the error and restart the program.
2A0-00B9	**Error message** Calculated error no. too large **Cause of error** Calculation of an error number for the FN14 function (ISO: D14) from a Q parameter resulted in a value outside the permissible range of 0 to 499. **Error correction** Edit the part program.
2A0-00BA	**Error message** Tool %s not defined **Cause of error** You called a tool that is not contained in the tool magazine. **Error correction** Check the pocket table and add the tool if required. A complete list of the tools that are not contained in the tool magazine can be found in the the TOOLLIST.ERR file stored in the TNC:\ directory.
2A0-00BB	**Error message** block scan inconsistent %s **Cause of error** During restoration of the machine status after a block scan (mid-program startup), the conditions on the machine regarding spindle speed (S), traverse range (R) or preset (P) were not the same as calculated in the block scan. **Error correction** - Shut down the control and restart it. - Inform your machine tool builder.

Error number	Description
2A0-00BC	**Error message** Limit switch %.2s- **Cause of error** The calculated path of the tool exceeds the traversing range (software limit switch) of the machine - Software limit switch was reached in a manual operating mode **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point. - Move the tool in the opposite direction.
2A0-00BD	**Error message** Limit switch %.2s- **Cause of error** The calculated tool path exceeds the machine's traverse limits. Negative traverse range is defined with MP92x.x. **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point.
2A0-00BE	**Error message** Limit switch %.2s+ **Cause of error** - The calculated path of the tool exceeds the traversing range (software limit switch) of the machine - Software limit switch was reached in a manual operating mode **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point. - Move the tool in the opposite direction.

Error number	Description
2A0-00BF	**Error message** Limit switch %.2s+ **Cause of error** The calculated tool path exceeds the machine's positive traverse limits. Positive traverse range is defined with MP91x.x. **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point.
2A0-00C1	**Error message** File does not exist **Cause of error** - The given file does not exist. - The given file has been deleted since last used. - You tried to select a file whose name exceeds the permitted length. - smarT.NC: You selected an .HU program that uses a point table that does not exist on the TNC hard disk. - You entered an incorrect path under >MOD >Print (print test). - You tried to open and edit a file protected by SELinux (access check). **Error correction** - Use an existing file for the selected file operation. - Pay attention to the maximum permissible length of the file name. - Check the .HU program for missing point tables and restore or read-in the missing files - Correct the path entry. - Deactivate SELinux temporarily.
2A0-00C2	**Error message** File type missing or incorrect **Cause of error** You tried to create a new file without first defining the appropriate file type. **Error correction** Enter the correct file type.

Error number	Description
2A0-00C3	**Error message** No datum table selected **Cause of error** You tried to select a datum number by soft key although no datum table is selected in the program header. **Error correction** - Under options in the program header (UNIT 700), enter a datum table from which the datums can be selected. - Make sure that you have entered the correct path name in the UNIT 141 or in the program header (UNIT 700).
2A0-00C7	**Error message** MC: System error in SPLC-RTS **Cause of error** - Internal software error in the run-time system (RTS) of the SPLC on the MC **Error correction** Inform your service agency
2A0-00C8	**Error message** Faulty recognition of PL/MB hardware **Cause of error** An error occurred during detection and evaluation of the safety-related PL and MB hardware. **Error correction** - Check the connection and function of HSCI-PLs and MB - Inform your service agency
2A0-00C9	**Error message** I/O assembly %1 reports an error **Cause of error** A safety-related HSCI-PL or the MB supplies invalid data. **Error correction** - Find and exchange the faulty device - Inform your service agency
2A0-00CA	**Error message** I/O assembly %1 reports an error **Cause of error** A safety-related HSCI-PL or the MB reports an error. **Error correction** - Check the wiring and condition of the PLs or the MB. - Inform your service agency

Error number	Description
2A0-00CB	**Error message** MC: System error in SPLC-RTS **Cause of error** - Internal software error in the run-time system (RTS) of the SPLC by the MC **Error correction** Inform your service agency
2A0-00CC	**Error message** MC: System error in SPLC-RTS **Cause of error** - Internal software error in the run-time system (RTS) of the SPLC by the MC **Error correction** Inform your service agency
2A0-00CD	**Error message** Run-time error in SPLC program **Cause of error** - Run-time error in the SPLC program **Error correction** Inform your service agency
2A0-00CE	**Error message** SPLC cannot load program **Cause of error** The SPLC program cannot be loaded. **Error correction** Inform your service agency
2A0-00CF	**Error message** SPLC program has changed **Cause of error** The SPLC program or the NC software was changed after the machine safety acceptance. **Error correction** Restore the original SPLC program or perform the safety acceptance again.
2A0-00D0	**Error message** SPLC program cannot run **Cause of error** SPLC program cannot be started **Error correction** - Note further error messages. - Inform your service agency

Error number	Description
2A0-00D1	**Error message** Machine not in safe operation **Cause of error** The control commissioning is not yet concluded. The functional safety of the machine is not ensured. **Error correction**
2A0-00D2	**Error message** Configuration of SPLC inputs **Cause of error** The configuration of the SPLC inputs with inverse logic is faulty. **Error correction** - Check configuration in safety-related machine parameters inpNoInverseA and inpNoInverseB. - Inform your service agency
2A0-00D3	**Error message** Configuration of SPLC inputs **Cause of error** The configuration of the SPLC inputs that are participating in the minute test is faulty. **Error correction** - Check configuration in safety-related machine parameter inpNoDynTest. - Inform your service agency
2A0-00D4	**Error message** Configuration of SPLC cycle time **Cause of error** Configured cycle time for SPLC is too long or too short. **Error correction** - Check configuration in machine parameter plcCount. - Inform your service agency
2A0-00D5	**Error message** Functional Safety (FS) not ensured! **Cause of error** This installation of the control software is a test version: - This control software may be used only for test purposes! - The functional safety of the machine is not guaranteed! - Therefore be careful when running the machine, particularly when the guard doors are open! **Error correction**

Error number	Description
2A0-00D6	**Error message** ACCESSLEVEL: Function locked **Cause of error** - Desired function in the active ACCESSLEVEL is locked **Error correction** - Enable the function through ACCESSLEVEL
2A0-00D8	**Error message** smarT.NC: Copy/Cut smarT.NC: Copy **Cause of error** **Error correction**
2A0-00D9	**Error message** Changed NC software version **Cause of error** - The NC software version was changed after the machine safety acceptance. - NC software version and version of the file used, SplcApi-Marker.def, do not match. **Error correction** - Transfer the SplcApiMarker.def file appropriate to the installed NC software version into the SPLC project. - Enter the value of the constant SPLC_API_VERSION from this file into the safety-related machine parameter splcApiVersion in CfgSafety - Repeat the safety inspection and approval of the machine with the appropriate comprehensiveness. - Inform your service agency
2A0-00DA	**Error message** Error in SPLC configuration data **Cause of error** - The configuration data for the SPLC are incorrect. It is not possible to translate the PLC program until these data are corrected. **Error correction** - Correct the configuration data for SPLC Note further error messages regarding this. - Inform your service agency

Error number	Description
2A0-00DB	**Error message** Incorrect condition of safe output %1 **Cause of error** - Indicated SPLC output is logical 1 (+24 V), although SPLC specifies logical 0 (0 V) for the output. - Faulty wiring (short circuit to +24 V) - PLD module defective **Error correction** - Check the wiring - Exchange the defective PLD module - Inform your service agency
2A0-00DC	**Error message** AFC: No reference power recorded **Cause of error** Adaptive Feed Control (AFC), learning mode: The TNC could not find any reference power. Possible cause: Test cut in the air **Error correction** Repeat the teach-in cut.
2A0-00DD	**Error message** Faulty assignment of tool/cut no. **Cause of error** AFC: In the AFC settings the assignment of the current cut number to the current tool is faulty. **Error correction** Repeat the teaching process. The TNC automatically resets the current cut to "Teach".
2A0-00DE	**Error message** Wiring of SPLC input %1 **Cause of error** According to the configuration (CfgSafety / inpNoDynTest), the SPLC input named in the text participates in the dynamic test but does not drop out although the corresponding test output was switched off. **Error correction** - Check the wiring - Check the configuration

Error number	Description
2A0-00DF	**Error message** Wiring of emergency stop **Cause of error** An emergency stop input does not drop out, although the associated test output was switched off. **Error correction** Check the wiring. All emergency stop circuits must be supplied with current over the corresponding test outputs.
2A0-00E0	**Error message** Wrong input %1 for dynamic test **Cause of error** The indicated SPLC input cannot be tested with the minute test (dynamic test) - No physical PLC input with the indicated number exists - The PLC input is neither on a system PL (PLB 62xxFS) nor on a safe machine operating panel (MB 6xxFS) even though there is more than one system PL in the HSCI system. **Error correction** - Check machine parameter inpNoDynTest in CfgSafety. - Inform your service agency.
2A0-00E1	**Error message** MC error during cross comparison at input %1 **Cause of error** The two terminals of a dual-channel FS input have differing logical conditions. Possible causes: - Key on the machine operating panel was pressed on a slant - Breakage in the wiring of an FS input - Short circuit on 0 V or 24 V in the wiring of an FS input - Incorrect configuration (SMP) of inverse FS inputs (e.g. with antivalent or inverted signal) **Error correction** - Check the keys on the machine operating panel. If one of the keys had been pressed at an incorrect angle, then no further measures are necessary - Check the wiring of the affected dual-channel input - Inform your service agency Note that the input cannot be put back into the trigger state until both input terminals are in the idle state.

Error number	Description
2A0-00E2	**Error message** Unexpected condition of SPLC input %1 **Cause of error** According to the machine parameter configuration, the given SPLC input in the idle state should be providing 0 V and another channel 24 V. This is not the case. Possible causes: - Configuration of the SPLC input is faulty - Input is not in the idle state during the self-test - Wiring is faulty. - The connected handwheel is not suitable for functional safety (FS) - The handwheel has been replaced by an unsuitable dummy plug (not suitable for FS) **Error correction** - Check the wiring - Check the configuration - Check connected handwheel and exchange it if necessary - Check the dummy plug and exchange it if necessary - Inform your service agency
2A0-00E3	**Error message** Select override: Handw./ Op. panel CAUTION F/S values **Cause of error** **Error correction**
2A0-00E4	**Error message** Control strategy not found in AFC.TAB. Default will be used. **Cause of error** The control could not find the control strategy assigned to the active workpiece in the table AFC.TAB. **Error correction** Correct the entries in the AFC column of the tool table or in the AFC.TAB table.
2A0-00E5	**Error message** Input marker %1 set by SPLC program **Cause of error** - The SPLC program has an input marker set to the value TRUE (= 1). This is not allowed. - Input markers can be deleted by the SPLC program (= 0), but they cannot be set (= 1) **Error correction** - Check the SPLC program and correct it if necessary - Inform your service agency

Error number	Description
2A0-00EC	**Error message** AFC parameters not allowed in turning mode **Cause of error** An AFC parameter was programmed that is not permitted in turning mode, e.g. TIME or DIST. **Error correction** Check the NC program and adapt it if necessary
2A0-00ED	**Error message** Program-run graphics: Incomplete display **Cause of error** The TNC is so busy machining the current workpiece that the program-run graphics are sometimes not being updated, and can therefore be incomplete. **Error correction** No corrective action possible
2A0-00EE	**Error message** AFC: No idle power recorded **Cause of error** The acceleration phase to the starting speed could not be concluded before the first cutting block was reached; therefore the idle power could not be determined. **Error correction** Only perform the first cutting block once it is ensured that the starting speed has been reached.
2A0-00EF	**Error message** AFC: function is not in effect; inconsistent input values **Cause of error** A FUNCTION MODE ... command was executed after the TOOL CALL. That is not allowed. **Error correction** Edit the NC program
2A8-0003	**Error message** Place handwheel in charger **Cause of error** The wireless handwheel is not located in the charging station although the handwheel mode is not active. If the rechargeable battery of the handwheel is emptied or there is interference in the radio connection, the TNC releases an emergency stop. In this case the program run is canceled. **Error correction** Always place the handwheel in the loading station when you are not working with it.

Error number	Description
2A8-0004	**Error message** Configuration error in OEM handwheel menu or soft key **Cause of error** The configuration of an OEM handwheel menu or an OEM handwheel soft key is incomplete or faulty. **Error correction** Inform your service agency.
2A8-0007	**Error message** Battery almost empty. Place handwheel into the charging station **Cause of error** The wireless handwheel's rechargeable battery is almost empty. If the rechargeable battery loses charge, it interrupts the connection to the handwheel. This results in an emergency stop and the program run is aborted! **Error correction** - Now place the handwheel onto the charging station to restore its charge. - Recommendation: Always keep the handwheel in the charging when not in use.
2A9-0001	**Error message** Faulty BLK FORM **Cause of error** The given workpiece blank definition is faulty and could not be interpreted by the control. **Error correction** - Correct the BLK FORM in the NC program.
2A9-0002	**Error message** Incorrect tool data **Cause of error** The 3-D simulation graphic cannot process the tool data. **Error correction** Adapt the tool data
2A9-0009	**Error message** Rapid-traverse cut near block %1 **Cause of error** The simulation detected possible cutting of material at rapid traverse near the indicated block number. **Error correction** - Check the position and size of the BLK FORM - Increase the quality of the simulation - Edit the NC program

Error number	Description
2A9-000B	**Error message** System error when calculating the graphics **Cause of error** System error during internal calculation of the 3-D graphical representation during program run or a test run. **Error correction** Inform your service agency
2A9-000C	**Error message** Violation of the workpiece near block %1 **Cause of error** The simulation detected a possible violation of the workpiece by a non-cutting part of the tool (holder or shank) near the indicated block number. **Error correction** Check the position and size of the BLK FORM, as well as the tool data, and edit the NC program if necessary
2A9-000D	**Error message** %2 3-D model not loaded %1 **Cause of error** Error while reading the 3-D model: the file could not be opened or is not a supported 3-D data format. **Error correction** - Check the path and correct it if necessary - Reload the file
2A9-000E	**Error message** %2 3-D model not loaded %1 **Cause of error** Could not load the 3-D model because it does not fulfill the quality requirements. The following requirements are in place for 3-D models: - All dimensions in mm - No gaps between triangles ("waterproof") - No overlapping - No degenerated triangles **Error correction** Regenerate the 3-D model and transfer it to the control.

Error number	Description
2A9-000F	**Error message** %2 3-D model not loaded %1 **Cause of error** Error while reading the 3-D model: the file contains too many triangles. **Error correction** - Use a rougher 3-D model - Generate the 3-D model with the CAD program again and transfer it to the control. In many CAD programs the level of detail can be set when exporting.
2A9-0010	**Error message** Workpiece could not be exported **Cause of error** Could not write the file. **Error correction** - Check the path and correct it if necessary - Check the available memory
2A9-0011	**Error message** Ext. workpiece monitoring temporarily deactivated **Cause of error** Too many blocks with violations of the workpiece were found. Extended workpiece monitoring will be deactivated until the next BLK FORM. **Error correction** Check and correct as necessary: - Tool data - Position and shape of the workpiece - Cuts at FMAX
2A9-0012	**Error message** Workpiece could not be exported **Cause of error** There is no workpiece present. **Error correction**
2AA-0005	**Error message** Database error **Cause of error** Access to the table was not possible for the following reasons: - The table is write-protected - The table is corrupted - The table does not exist **Error correction** Check the table

Error number	Description
2D4-0000	**Error message** Start of Python script "%1" failed **Cause of error** Communication with the PLC necessary for starting the Python script has failed. **Error correction** - Compile the PLC program - Restart the Python script
2D4-0001	**Error message** Start of Python script "%1" failed **Cause of error** The exact cause of the error is unknown. **Error correction** Inform your service agency.
2D4-0002	**Error message** Start of Python script "%1" failed **Cause of error** The Python option is not enabled For Python applications to be run on the control, the "Python OEM Process" software option must be enabled in the SIK. **Error correction** - "Python OEM process" software option was not enabled in the SIK
2D4-0003	**Error message** Start of Python script "%1" failed **Cause of error** Too little free working memory available to run the Python script. **Error correction** - Assign less memory requirement to the Python script over the machine configuration ("memLimit" machine parameter) - Close running Python processes to free memory.
2D4-0004	**Error message** Start of Python script "%1" failed **Cause of error** The value of the machine parameter "memLimit" (maximum process memory) is invalid. **Error correction** - Check the machine parameter "memLimit" and correct it

Error number	Description
2D4-0005	**Error message** Start of Python script "%1" failed **Cause of error** The path entered in the machine configuration for the Python script is invalid. **Error correction** - Correct the machine parameter in the CfgSoftkeyOverlay config object.
2D4-0006	**Error message** Start of Python script "%1" failed **Cause of error** The name entered in the machine configuration for the Python process is invalid. **Error correction** - Correct the machine parameter "jobName" in the CfgSoftkeyOverlay config object.
2D4-0007	**Error message** Start of Python script "%1" failed **Cause of error** A Python process with the same name is already running. **Error correction** - If desired, terminate the running Python process
2D4-0008	**Error message** Start of Python script "%1" failed **Cause of error** In the machine configuration there are invalid parameters defined for the Python script. **Error correction** - Correct the machine parameter "parameter" in the CfgSoftkeyOverlay config object.
303-0001	**Error message** Referenced OPC UA namespace %1 does not exist **Cause of error** The entities CfgOpcUaObject and CfgOpcUaPlcVar with the following keys do not refer to an existing OPC UA namespace configuration: %1 The nodes stated and all child elements will not be created. **Error correction** Check the withinNamespace attribute

Error number	Description
303-0002	**Error message** Referenced OPC UA parent node %1 does not exist **Cause of error** The entities CfgOpcUaObject and CfgOpcUaPlcVar with the following keys use the attribute parentNode to refer to a parent node that does not exist: %1 The nodes and all child elements will not be created. **Error correction** Check the parentNode attribute
303-0003	**Error message** Invalid value for nodeIdIdentifier %1 **Cause of error** nodeIdType was set to Numeric in the configuration datum %1. However, the value entered for nodeIdIdentifier is not a numeric value. The node and all child elements will not be created. **Error correction** Check the nodeIdType and nodeIdIdentifier attributes in the indicated configuration datum
303-0004	**Error message** Invalid value for publicationDate %1 **Cause of error** The publication date entered in the configuration datum %1 does not match the format YYYY-MM-DDThh:m-m:ss.sssTZD. **Error correction** Check the configuration datum
303-0005	**Error message** Multiple definitions of the namespace URI %1 **Cause of error** The namespace URI of an OPC UA namespace must be unique. The namespaceUri in the CfgOpcUaNamespace entities with the following keys is identical: %1 (%2 entities) Only the first namespace configuration stated is active. **Error correction** Check the configuration of the namespaceUri

Error number	Description
303-0006	**Error message** Value not permitted for namespaceUri %1 **Cause of error** The OPC UA namespace URI entered in the configuration datum %1 is not permitted, since it is too similar to other names or namespace URIs that have already been assigned. The namespace and all nodes it contains will not be created. **Error correction** Select a different URI for the OPC UA namespace
303-0007	**Error message** Multiple use of NodeId %1 **Cause of error** The same NodeId was configured for the entities CfgOpcUaObject and CfgOpcUaPlcVar with the following keys: %1 (%2 entities) Only the first node stated will be created. **Error correction** Check the configuration of the withinNamespace, nodeId-Type, and nodeIdIdentifier attributes
303-0008	**Error message** Attribute %1 was not configured **Cause of error** No value was entered in the configuration datum %1. It is absolutely essential. The node and all child elements will not be created. **Error correction** Check the configuration datum
303-0009	**Error message** Cyclic parent-child relationship **Cause of error** The entities CfgOpcUaObject and CfgOpcUaPlcVar with the following keys form a cyclic reference though their parentN-ode attributes: %1 This is not permitted. The nodes stated and all child elements will not be created. **Error correction** Check the respective parentNode attributes

Error number	Description
303-000A	**Error message** Invalid namespace URI %1 **Cause of error** An invalid namespace URI was entered in the namespaceUri attribute in the configuration datum %1. The node and all child elements will not be created. **Error correction** Enter a correct namespace URI
303-000B	**Error message** Invalid parent node: %1 **Cause of error** The entity CfgOpcUaObject %1 references an CfgOpcUaPlcVar entity as parent node. This is not allowed. The node and all child elements will not be created. **Error correction** Check the parentNode attribute
303-000C	**Error message** Impermissible multiple use of browseName %1 **Cause of error** The same browseName is used below the same parent node in the entities CfgOpcUaObject and CfgOpcUaPlcVar with the following keys: %1 (%2 entities) Only the first node stated will be created. **Error correction** Check the configuration of the browseName attribute
303-000D	**Error message** More than %1 variables are configured **Cause of error** The machine manufacturer configured many variables for access through OPC UA NC Server. If OPC UA clients order subscriptions to all of these variables, a system overload can occur. %2 variables are configured. **Error correction** Reduce the number of variables

Error number	Description
303-000E	**Error message** The configuration of the OPC UA NC Server has changed **Cause of error** The OEM-specific configuration of OPC UA NC Server was changed. The changes will take effect the next time the server is restarted. **Error correction** Restart the server to activate the changes. Active connections will be disconnected.
303-0012	**Error message** Invalid range specified %1 **Cause of error** The range specified in the configuration datum %1 is not valid. The minimum value is greater than the maximum value. The variable node will not be created. **Error correction** Check the configuration datum
303-0013	**Error message** Incomplete range specified %1 **Cause of error** The range specified in the configuration datum %1 is not complete. Either the minimum value or the maximum value is missing. The variable node will not be created. **Error correction** Check the configuration datum
303-0014	**Error message** EURange outside of InstrumentRange **Cause of error** The range specified for EURange in the configuration datum %1 exceeds the range specified for InstrumentRange in the configuration datum %2 InstrumentRange indicates the maximum permissible range of values, and therefore must not be exceeded by EURange. The variable node will not be created. **Error correction** Check the configuration datum

Error number	Description
303-0015	**Error message** Unknown UnitCode %1 **Cause of error** The UnitCode entered in the configuration datum %1 is unknown to the control. The variable node will not be created. **Error correction** Check the configuration datum
303-0016	**Error message** Unknown PLC symbol %1 **Cause of error** Im Konfigurationsdatum %1 wurde ein PLC-Symbolname angegeben, der im PLC-Programm nicht definiert ist. **Error correction** Konfigurationsdatum prüfen
303-0017	**Error message** Meta-information does not match the PLC operand **Cause of error** In Konfigurationsdatum %1 wurden Meta-Informationen für numerische Werte konfiguriert. Der Datentyp des angegebenen PLC-Operanden ist jedoch nicht numerisch. Die Meta-Informationen werden ignoriert. **Error correction** Meta-Informationen löschen oder korrekten PLC-Operanden angeben
303-0018	**Error message** Meta-information does not match the PLC operand **Cause of error** A valuePrecision was entered in the configuration datum %1. However, this is supported only for the data types Word and DWord. The indicated valuePrecision will be ignored. **Error correction** Delete the value for valuePrecision or enter a correct PLC operand

Error number	Description
303-0019	**Error message** Invalid index for PLC symbol %1 **Cause of error** Im Konfigurationsdatum %1 wurde ein PLC-Symbol mit einem ungültigen Index adressiert. **Error correction** Konfigurationsdatum prüfen
303-001A	**Error message** PLC symbol %1 has no valid data value **Cause of error** Im Konfigurationsdatum %1 wurde ein PLC-Symbolname angegeben, der keinen gültigen Datenwert beschreibt (z.B. eine PLC-Struktur). **Error correction** Konfigurationsdatum prüfen. (Bei PLC-Strukturen können deren Elemente als separate Variablen konfiguriert werden.)
303-001B	**Error message** Write-access configured for PLC constant **Cause of error** In Konfigurationsdatum %1 wurde der PLC-Operand über valueWritable als schreibbar konfiguriert. Der angegebene PLC-Operand ist jedoch eine Konstante und kann somit nicht geändert werden. Das Attribut valueWritable wird ignoriert. **Error correction** Konfigurationsdatum prüfen
320-0001	**Error message** Error while sending message to PLC **Cause of error** **Error correction**
320-0002	**Error message** PLC program not successfully compiled **Cause of error** **Error correction**

Error number	Description
320-0003	**Error message** Argument out of range **Cause of error** **Error correction**
320-0004	**Error message** PLC is running in simulation mode **Cause of error** **Error correction**
320-0005	**Error message** PLC compiler argument (%1) missing **Cause of error** **Error correction**
320-0006	**Error message** Invalid PLC call parameter **Cause of error** **Error correction**
320-000A	**Error message** Programmed spindle speed is too low **Cause of error** The spindle speed that programmed is too low. **Error correction** Program a faster shaft speed or check the configuration datum Axes->ParameterSets->????->CfgFeedLimits->minFeed. "????" designates the current name of the data record from the configuration.
320-000B	**Error message** Rotational speed programmed for spindle too high **Cause of error** The shaft speed that you programmed for this axis is too low. **Error correction** Program a faster shaft speed or check the configuration datum Axes->ParameterSets->????->CfgFeedLimits->minFeed. "????" designates the current name of the configuration set.

Error number	Description
320-000C	**Error message** %1 %1 **Cause of error** siehe Maschinenhandbuch **Error correction** siehe Maschinenhandbuch
320-000D	**Error message** The PLC program has been stopped **Cause of error** The PLC program was stopped because of a system error in the PLC. **Error correction** Inform your machine tool builder.
320-000E	**Error message** Error in PLC program **Cause of error** An error occurred during execution of the PLC program. **Error correction** Inform your machine tool builder.
320-000F	**Error message** System error in the PLC System error in the PLC **Cause of error** Internal software error **Error correction** Inform your service agency.
320-0010	**Error message** General system error **Cause of error** Internal software error **Error correction** Inform your service agency.
320-0011	**Error message** Spindle speed programmed without configured spindle **Cause of error** No spindle available, but speed has been programmed. **Error correction** Configure the spindle (CfgAxes->spindleIndices) or do not program a spindle speed.

Error number	Description
320-0012	**Error message** Configuration datum %1/%2 contains errors Configuration datum %1 - %2 contains errors **Cause of error** The given configuration datum contains errors and was not accepted for control operation. **Error correction** Correct the given configuration data or inform your machine tool builder.
320-0013	**Error message** Configuration datum %1 for channel %2 missing **Cause of error** The given configuration datum was not found. **Error correction** Add the given configuration datum or inform your machine tool builder.
320-0014	**Error message** Configuration datum %1 for channel %2 already defined **Cause of error** The given configuration datum causes a multiple definition of an output to the PLC. **Error correction** Correct the given configuration data or inform your machine manufacturer.
320-0015	**Error message** Operand %1 for configuration datum %2 not found **Cause of error** The operand described in the given configuration data was not defined in the PLC. **Error correction** Correct the given configuration data, select another PLC program and/or inform your machine manufacturer.

Error number	Description
320-0016	**Error message** Data of PLC strobe for channel %1 could not be transferred **Cause of error** The data belonging to the given output to the PLC could not be saved in PLC markers. **Error correction** Correct the associated configuration datum or inform your machine tool builder.
320-0017	**Error message** Configuration and PLC program incompatible **Cause of error** A PLC program has been selected that is not compatible with the configuration of the machine. Necessary prerequisites for the successful operation of the PLC program: - PLC programs that operate with the numerical memory interface API 1.0 (TNC flag interface) can only control one operating mode group, one NC channel, and one spindle. - PLC programs that use the symbolic memory interface API 3 must create the API data structures in the number suitable for the configuration. - The correct values for the system parameters AXIS_COUNT and SPINDLE_COUNT must be defined in the configuration file of the PLC compiler. - PLC programs must define a sufficiently large memory image. All permanently assigned, non-volatile, or absolutely addressed flags must fall within the memory image. - PLC programs have to define all markers to which configuration data are to be assigned at the start of the PLC program. - Pay attention to any other, previously issued warnings. **Error correction** - Check the machine configuration - Check the configuration file of the PLC compiler for correct values - Use a suitable PLC program - Inform your service agency
320-0018	**Error message** Inconsistent PLC program **Cause of error** The API version in the selectable PLC program is not compatible with the control software or the definition of the API version in the PLC program is faulty. **Error correction** Update the ApiMarker.DEF file from the PLC project of the machine, or correct the PLC program.

Error number	Description
320-0019	**Error message** Error in PLC program: Input was changed **Cause of error** The PLC program tried to change an input marker (e.g. I3). **Error correction** Correct the PLC program. Input markers can only be read, never written.
320-001A	**Error message** PLC error table cannot be read (%1) **Cause of error** The PLC cannot read your error table. It could be that the path configured for the table is incorrect or the table has an incorrect format. **Error correction** Check the configuration and PLC error table.
320-001B	**Error message** PLC error table erroneous (%1) **Cause of error** An invalid error marker was given in the PLC error table.It could be that a symbolic name is written incorrectly. **Error correction** Correct the PLC error table.
320-001C	**Error message** Control cannot read operating times **Cause of error** The control cannot read one or more operating time values that are saved in a file. The file was presumably destroyed. **Error correction** If the error message appears repeatedly, inform your service agency.
320-001D	**Error message** System error in the PLC **Cause of error** Internal software error: The persistent storage of operating times in a file has failed, presumably because of an error in the file system. **Error correction** Inform your service agency.

Error number	Description
320-0023	**Error message** Invalid configuration for fast inputs Number %1 - Operand %2 **Cause of error** The configuration of the fast inputs is incorrect. **Error correction** Correct the configuration under consideration of the following constraints: - PLC programs that use the TNC marker can only use the markers M4590-4593 - The data type D (DWORD) is required to record all edges - Symbolic operands must be defined in the PLC program
320-0024	**Error message** Configuration datum %1/%2 missing **Cause of error** The given configuration datum was not found. **Error correction** Add the given configuration datum or inform your machine tool builder.
320-0025	**Error message** The PLC program has been stopped **Cause of error** Due to a change of the configuration data, the PLC was stopped and will be restarted. **Error correction** No correction necessary.
320-0026	**Error message** PLC: Division by 0/ Modulo error **Cause of error** Run-time error in the PLC program: - Division by 0 was caused. - A modulo calculation was incorrectly executed. **Error correction** Edit the PLC program. Change the PLC compiler setting DIVERROR/MODERROR.
320-0027	**Error message** PLC: Overflow when multiplying **Cause of error** Run-time error in PLC program: - Overflow during multiplication **Error correction** Edit the PLC program. Change the PLC compiler setting MULERROR.

Error number	Description
320-0028	**Error message** Invalid configuation for PLC arithmetic error operand %1 **Cause of error** The configuration for treatment of PLC arithmetic errors is illegal. **Error correction** Correct the configuration under consideration of the following constraints: - Examine the MULERROR, DIVERROR, and MODERROR entries in the configuration file for the PLC compiler (see entry under CfgPlcPath) - PLC programs that use the TNC marker interface can use only the markers M4200 to M4202 - Symbolic operands must be defined in the PLC program
320-0029	**Error message** System error in the PLC **Cause of error** A PLC server function that has not yet been implemented was called. **Error correction** Inform your service agency
320-002A	**Error message** System error in the PLC **Cause of error** The PLC server cannot find the sender of a message. **Error correction** Inform your service agency
320-002B	**Error message** System error in the PLC **Cause of error** The PLC server cannot reach the sender of a message. **Error correction** Inform your service agency
320-002C	**Error message** System error in the PLC **Cause of error** A software error has occurred in the PLC. **Error correction** Inform your service agency

Error number	Description
320-002D	**Error message** Cannot read the PLC MAIN file (%1) **Cause of error** The PLC cannot read its program file. **Error correction** Check the configuration and PLC program file.
320-002E	**Error message** Cannot read the PLC compiler file (%1) **Cause of error** The PLC cannot read a given compiler file. **Error correction** Check the PLC compiler configuration file (.cfg).Check the PLC compiler error text file (.err).
320-002F	**Error message** Operand field %1 in datum %2 too large **Cause of error** The given symbolic name refers to a field of PLC operands. The field size is too large for the given configuration data item. **Error correction** Correct the given configuration item, or contact your machine tool builder.
320-0030	**Error message** Operand field %1 in datum %2 too small **Cause of error** The given symbolic name refers to a field of PLC operands. The field size is too small for the given configuration data item. **Error correction** Correct the given configuration item, or contact your machine tool builder.
320-0031	**Error message** Invalid text source **Cause of error** A text source configured for dialog and error texts cannot be loaded. **Error correction** Configure another language or contact your machine tool builder.

Error number	Description
320-0032	**Error message** PL510 fault in cyclic operation **Cause of error** An EMC disturbance has occurred in the cyclical PL 510 mode. The periphery cannot be used. **Error correction** Inform your machine tool builder
320-0034	**Error message** Strobe was not realized **Cause of error** A strobe output was not realised after the block scan **Error correction**
320-0035	**Error message** System-management-bus initialization failed **Cause of error** No support for system management bus, or version of operating system is too old. **Error correction** Inform your service agency.
320-0036	**Error message** Error in PLC Python script **Cause of error** The given PLC Python script has an error. **Error correction** Inform your machine tool builder.
320-0037	**Error message** Error in PLC program **Cause of error** PLC run-time error: An error occurred when an API was called in the PLC program. **Error correction** Edit the PLC program.

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed parameter is outside the valid range **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed parameter is invalid/does not exist **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed address is outside the valid value range **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Sum of address and block length outside valid value range **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed address is not a word or double word address **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed value can/must not be changed **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Programmed file is faulty **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Wrong NC operating mode is active **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - A position command or another job is already being executed **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - No tool changer defined **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - String cannot be converted or contains illegal characters in the string **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - No end of string detected, or string incomplete **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - No connection over interface or to a server **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - RS-232 interface busy or not assigned **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - RS-232 transmit buffer not empty **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - RS-232 receiving buffer empty **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - RS-232 baud rate not possible **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Transmission error of RS-232 interface **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - No current controller found **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The PLC module was not called as a submit or spawn job **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** An error occurred when a PLC module was called in the PLC program. The PLC module was called during program run without a strobe. **Error correction** - Check and correct the PLC program - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Overflow of key queue **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Overflow of PLC error message queue **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The PLC module was called from a submit or spawn job **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Too many elements in a constants field **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Illegal elements programmed in a constants field **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Oriented spindle stop already active **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Module function is already active **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The entered file name is invalid! **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed field name does not exist **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Syntax error of the query statement **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - No fitting data record available for the inquiry **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed axis has not yet been referenced **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The reset of a connected external device is faulty **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The editor is not active **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - A general error occurred during a data access **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The control's system memory is too small **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Error occurred during parsing **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Fast PLC input set when drives are to be switched on **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The drives cannot be switched on **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The inspection of the number type was interrupted during entry in the tool table **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The powerfail monitoring cannot be switched off **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The control is a system without functional safety (FS) **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Error occurred in the evaluation of the soft-key resource file **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The PLC module execution was canceled **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - There has been an Profibus error **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program **Error correction** - Check and correct the PLC program - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed function is not supported by the PLC module **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed function is supported only by a dual-processor control. **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The control hardware does not have a serial interface **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The programmed function is not supported by a dual-processor control **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - File could not be disabled **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Entered file not found **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The function (software option or FCL) has not not been released **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The function not available when there is more than one configured NC channel **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Could not start the programmed process (Python) **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - This hardware does not support the programmed analog output at X8/X9 **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The PLC module could not be run **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The transferred symbol/designator does not exist **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Could not create the handle **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - The transferred handle is invalid **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - Wrong data direction for transmission **Error correction** - Check the PLC program and correct it - Inform your service agency

Error number	Description
320-0037	**Error message** Error in PLC program **Cause of error** - An error occurred when an PLC module was called in the PLC program - A control with functional safety (FS) does not support the function called **Error correction** - Check the PLC program and correct it - Inform your service agency
320-0038	**Error message** Operand %1 for configuration datum %2 illegal **Cause of error** The operand defined in the given configuration datum is not allowed at this location. In most cases, configuration data can refer only to ordinary logical PLC markers (type M) or ordinary arithmetic PLC operands (types B,W,D). **Error correction** Correct the given configuration datum, and select another PLC program or inform your machine manufacturer
320-0039	**Error message** Configuration datum %1 **Cause of error** Correct the given configuration datum or inform your machine manufacturer **Error correction** Too many configuration data of the given type were defined.
320-003A	**Error message** PLC: Error in magazine rules file **Cause of error** The selected magazine rules file is missing or contains errors. **Error correction** Restore or correct the magazine rules file.

Error number	Description
320-003B	**Error message** Default setting of IOC hardware is incomplete **Cause of error** The default setting for the use of the IOC hardware and the IOC configuration file is incomplete: (see also the machine parameter iocProject under CfgPlcPeriphery) - IOC file not found - Hardware is configured in the IOC file, but the file is not available - IOC hardware is available, but the IOC file is not configured **Error correction** - Match the configuration file and hardware. - You can find more information in the diagnostics menu
320-003C	**Error message** Profibus initialization incorrect **Cause of error** An error occurred during initialization of the Profibus hardware. **Error correction** - Inform your service agency - You can find more diagnostic information in the diagnostics menu.
320-003D	**Error message** Profibus error in cyclic operation **Cause of error** An error occurred during access to the Profibus hardware. **Error correction** - Inform your service agency - You can find more diagnostic information in the diagnostics menu.
320-003E	**Error message** Faulty initialization of IOC hardware **Cause of error** Errors occurred while initializing the IOC hardware and the IOC configuration file (see the attribute iocProject under CfgPlcPeriphery). **Error correction** - Inform your service agency. - You can find more information in the diagnostics menu.

Error number	Description
320-003F	**Error message** Faulty HSCI initialization **Cause of error** An error occurred during initialization of the HSCI hardware. **Error correction** - Inform your service agency - You can find more diagnostic information in the diagnostics menu.
320-0040	**Error message** Error from PL/MB in cyclic operation **Cause of error** A PL component (PLB), a machine operating panel (MB, TE), or a PL part of a compact inverter (UEC, UMC) reported an error. **Error correction** - Inform your service agency - For more diagnostics information about the affected device and the cause, see the bus diagnosis under HSCI
320-0041	**Error message** SPLC MAIN file cannot be read (%1) **Cause of error** The PLC cannot read the program file for the SPLC program. **Error correction** Check configuration and SPLC program file.
320-0042	**Error message** Inconsistent SPLC program **Cause of error** The API version in the selectable SPLC program is not compatible with the control software or the definition of the API version in the SPLC program is faulty. **Error correction** Update the file SplcApiMarker.DEF or correct the SPLC program. The file SplcApiMarker.def must be included in the SPLC program before all other definition files.
320-0043	**Error message** Inconsistent SPLC program **Cause of error** The definition of the markers that are transferred between the PLC program and SPLC program is faulty. **Error correction** Correct the SPLC program

Error number	Description
320-0044	**Error message** SPLC program has changed **Cause of error** The SPLC program has been edited. A partial acceptance test must be conducted before the machine can be operated with this edited SPLC program. **Error correction** Intended change: Delete the error message, then restart the control and run the required partial acceptance test. Unintended change: Undo the change of the SPLC program and recompile the SPLC program.
320-0045	**Error message** SPLC program binary code has changed **Cause of error** The binary code of the SPLC program has been changed, although the source code has remained unchanged. Possible causes: - New SPLC compiler through software update - Corrupted binary file of the SPLC program on MC or CC (without software update) **Error correction** Message appears after a software update: Delete the error message, then restart the control and run the required partial acceptance test. Then take the new CRC sum into the corresponding safe machine parameters. Message appears, although no software update has been conducted: Inform your service agency
320-0047	**Error message** Configuration PLC compiler: Enter %1 **Cause of error** A value inappropriate for the control model is defined for a constant in the configuration file for the PLC compiler. The values of the constants OMG_COUNT, CHANNEL_COUNT, AXIS_COUNT and SPINDLE_COUNT must be entered correctly. **Error correction** - Check the configuration file of the PLC compiler and correct it if necessary

Error number	Description
320-0048	**Error message** PLC program source code required **Cause of error** Because of a reconfiguration, the PLC program is no longer suitable on the control. The PLC program must be regenerated from the source code. **Error correction** - Undo the configuration change (ignore the repeated error message). - Copy the source code of the PLC program to the control and compile it.
320-004C	**Error message** Error: Process %1 could not be started **Cause of error** The hardware used does not have enough main memory. **Error correction** Inform your service agency.
320-004D	**Error message** Error: Process %1 could not be started **Cause of error** The command line transferred to the process is too long. **Error correction** Ensure that the command line has fewer than 127 characters.
320-004E	**Error message** Error: Process %1 could not be started **Cause of error** The name selected for the process is ambiguous. **Error correction** Select another unique name for the process.

Error number	Description
320-004F	**Error message** Error: Process %1 could not be started **Cause of error** Possible causes: - The given path does not lead to a valid Python script. - Fatal errors occurred during initialization of the Python script. - The assigned memory was exhausted during initialization of the Python script. - Other system resources were exhausted during initialization of the Python script. **Error correction** - Check the path of the Python script and correct it if necessary. - Ensure that all required libraries are installed in the required versions. - Increase the memory reserved for the script.
320-0050	**Error message** Error: Process %1 could not be started **Cause of error** An excessively long name was entered for the Python process. **Error correction** - Edit the Python script. The name of the Python process must be shorter than 17 characters.
320-0051	**Error message** Error: Process %1 could not be started **Cause of error** Possible causes: - The path given for the Python script is too long. - No file was found in the path given for the Python script. **Error correction** - Ensure that a Python script has been saved in the given path. - Check the path information. The path must have fewer than 260 characters.

Error number	Description
320-0052	**Error message** Error: Process %1 could not be started **Cause of error** Possible causes: - You have assigned too much memory to the Python script. - You have assigned more memory to the script than is available in total for all Python processes. - You have assigned a negative value to the Python script. - You have not assigned any memory to the Python script. **Error correction** - Adapt the Python script so that it is assigned a correct memory size.
320-0053	**Error message** Error: Process %1 could not be started **Cause of error** The maximum memory available for Python processes is in use. **Error correction** - Edit the Python scripts so that fewer processes are started at the same time. - Assign less memory for the individual processes.
320-0054	**Error message** Error: Process %1 could not be started **Cause of error** Could not start the Python process because it would exceed the maximum number of simultaneous Python processes. **Error correction** - Edit the Python scripts so that fewer processes are active at the same time.
320-0055	**Error message** Error: Process %1 could not be started **Cause of error** The software option #46 (Python OEM Process) required for execution of Python processes is not enabled. **Error correction** - Order the software option from your service agency.
320-0056	**Error message** Initialization of IOC hardware with warnings **Cause of error** Warning occurred during the initialization of IOC hardware and the IOC configuration file (see the parameter iocProject under CfgPlcPeriphery). **Error correction** You can find more information in the diagnostics menu.

Error number	Description
320-0057	**Error message** ProfiNET: Initialization faulty **Cause of error** An error occurred during initialization of the ProfiNET hardware. **Error correction** - Inform your service agency. - You can find more information in the diagnostics menu.
320-0058	**Error message** ProfiNET: Error in cyclic operation **Cause of error** An error occurred during access to the ProfiNET hardware. **Error correction** - Inform your service agency. - You can find more information in the diagnostics menu.
320-0059	**Error message** IOC configuration has invalid PLC operand addresses **Cause of error** The IOC configuration contains PLC operand addresses, that exceed the permissible PLC memory area. Possible causes: - IOC configuration and PLC configuration file do not fit together - Initial servicing or new installation - Importing a backup The BUS diagnosis shows which operand range is affected. **Error correction** Confirm the error message and current interruption, and note the further error messages: - Either the areas for the affected operands must be enlarged in the configuration file for the PLC compiler, or - a restart is necessary in order to activate a new or changed configuration. - Inform your service agency
320-005A	**Error message** IOC configuration requires too much memory **Cause of error** - The IOC configuration requires for at least one bus system (e.g. HSCI) too much memory on the field bus hardware. - The BUS diagnosis has more information. **Error correction** - Check the IOC configuration. - Reduce the number of configured system components. - Create a service file. - Inform your service agency.

Error number	Description
320-005B	**Error message** HSCI warning in cyclic operation **Cause of error** An warning appeared during access to the HSCI hardware. **Error correction** - You can find more diagnostic information in the HSCI-bus diagnostics.
320-005C	**Error message** Change of %1/%2 requires a compilation of the PLC program **Cause of error** The changed configuration datum goes into the definition of symbols for the PLC program. The PLC program must be recompiled so that the change becomes effective. **Error correction** Recompile the PLC program
320-005D	**Error message** Internal inputs/outputs: Faulty initialization **Cause of error** An error occurred during initialization of the internal I/O hardware (PL/PL510/SPI). **Error correction** - Inform your service agency. - You can find more information in the diagnostics menu
320-005E	**Error message** Internal inputs/outputs: Error in cyclic operation **Cause of error** An error occurred upon access to the internal I/O hardware (SPI modules). **Error correction** - Inform your service agency. - You can find more information in the diagnostics menu.
320-005F	**Error message** Spindle not available **Cause of error** An NC function is trying to control a spindle that is not available at the moment: - At present the spindle is functioning as an NC axis - The spindle is occupied by another channel - A spindle is to be switched to the NC axis while it is occupied by the NC **Error correction** Edit the NC program or contact your machine tool builder

Error number	Description
320-0060	**Error message** IOC configuration has invalid PLC operands **Cause of error** The IOC configuration includes PLC operands that have already been assigned. The BUS diagnosis shows which operands are affected. Possible causes: - Use of operands that are occupied by an internal PL board. - Use of operands that are occupied by a handwheel. **Error correction** Correct the operand addresses in the IOC file: - If an internal PL board is active, then the operands I0-I32 and O0-O31 must not be used. - If the default data of a handwheel are active, then the operands I160-I175 and O96-O111 must not be used. - Inform your service agency.
320-0061	**Error message** Invalid IOC configuration for PLC operand Operand %1 not found **Cause of error** The operand designated in the IOC file was not defined in the PLC program. **Error correction** - Correct the PLC program or the IOC project. - Inform your service agency.
320-0062	**Error message** Invalid IOC configuration for PLC operand Operand %1 not appropriate for terminal (%2) **Cause of error** The operand designated in the IOC file is not suitable for the terminal: - The data size is incorrect. - The terminal cannot be mapped to the operand type. - Mixture of logical and arithmetical data types. **Error correction** - Correct the PLC program or the IOC project. - Inform your service agency.
320-0063	**Error message** Invalid configuration for PLC log files **Cause of error** The configuration under CfgPlcLogging is invalid. **Error correction** Inform your service agency

Error number	Description
320-0064	**Error message** Strobe %1 was canceled **Cause of error** Execution of the NC program was canceled in the output of an M, S, or T strobe to the PLC program. **Error correction** - Your system might be in an inconsistent condition. Check whether the status shown for the machine and for the active tool reflect the actual situation. - Inform your service agency.
320-041A	**Error message** PLC: excessive nesting **Cause of error** PLC runtime error: - You attempted to nest more than 32 module calls within each other. - You programmed a recursive module call that exceeds the limit of 32 levels. **Error correction** Edit PLC program.
320-041B	**Error message** PLC: stack underflow **Cause of error** PLC runtime error: You attempted to retrieve data from the stack, although it had not been written there. **Error correction** Edit PLC program.
320-041C	**Error message** PLC: stack overflow **Cause of error** PLC runtime error: You attempted to write more than 128 bytes of data to the stack. Word operands (B/W/D/K) each occupy 4 bytes, Logic operands (M/I/O/T/C) occupy 2 bytes. **Error correction** Edit PLC program.

Error number	Description
320-041D	**Error message** PLC: timeout PLC: timeout **Cause of error** PLC runtime error: - The processing of the cyclically executed program section takes too long. Check the subprogram structure for very calculation-intensive sections that you can start as SUBMIT jobs. - The displayed processing time will be increased during data transfer and in handwheel mode. In case of doubt, select handwheel mode and simultaneously start the data transfer at max. baud rate. At the same time, check "MAXIMUM PROCESSING TIME" in the PLC programming. Values should not exceed 150% (safety reserve in the event of unfavorable operating conditions!). **Error correction** Edit the PLC program.
320-041E	**Error message** PLC: CASE out of range **Cause of error** PLC runtime error: The operand for the CASE statement contains a value that cannot be interpreted as an offset in the CM table (smaller than 0, or greater than or equal to the table length). **Error correction** - Have the PLC program checked - Inform your service agency
320-041F	**Error message** PLC: subprogram not defined **Cause of error** PLC runtime error: Subprogram not defined. **Error correction** Edit the PLC program.

Error number	Description
320-0420	**Error message** PLC: index range incorrect **Cause of error** PLC runtime error: - The address for a writing access to data types B/W/D/M/I/O/T/C is, through the inclusion of the index register, in an invalid region for these operand types. - The index register contains a value, due to accessing a constant field, which is not possible for this field (less than 0, or greater than or equal to field length). - The address of a string leads through the inclusion of the index register to an invalid value. - The number of a dialog (S#Dn[X]) or an error message (S#En[X]) leads through the inclusion of the index register to an invalid value (less than 0 or greater than 999). - During the addressing of a component string. **Error correction** Edit the PLC program.
320-0421	**Error message** PLC: error table missing **Cause of error** There is no PLC error table. - A PLC error module 9085/9086 was called although no error table was compiled, or there were no entries in the table. - A PLC error module 9085/9086 was called or an error marker was set, although the error table was edited or erased after compilation. **Error correction** - Compile the PLC error table. - Check the entries in the PLC error table.
320-0422	**Error message** PLC: error in module call **Cause of error** Fatal error during PLC module call (e.g. Module 9031: error converting MP). **Error correction** Edit the PLC program.
320-0423	**Error message** PLC: Event file not found **Cause of error** In the system file plc.cfg the file defined with PLCEVENTS= was not found. **Error correction** Contact your service agency.

Error number	Description
320-0424	**Error message** PLC: Too many events **Cause of error** More than 15 events were defined for the current SPAWN process (cooperative multitasking). **Error correction** Contact your service agency.
320-0442	**Error message** PLC: error table not .PET **Cause of error** The PLC error table selected in plc.cfg is not a PET file. **Error correction** Check the format of the PLC error table.
320-0443	**Error message** PLC: error table not found **Cause of error** The PLC error table in the plc.cfg file was not found. **Error correction** Check the file name or the path name.
320-0444	**Error message** PLC: err. table format incorrect **Cause of error** PLC error table: The error table selected in the plc.cfg file does not have an up-to-date binary format (e.g. after a software exchange). **Error correction** Delete the PLC error table and download a new PLC error table through the data interface.
320-0445	**Error message** PET table: Too many lines **Cause of error** There are too many error messages defined in the PET table. **Error correction** Only a limited number of lines in a PET table can be evaluated. All subsequent lines are ignored.
320-07D0	**Error message** PLC: Checksum error **Cause of error** **Error correction**

Error number	Description
320-07D1	**Error message** PLC: M4005/M4006/M4006 incorrect **Cause of error** PLC runtime error: More than one of the markers M4005 (M03), M4006, (M04), M4007 (M05) is set. **Error correction** Edit the PLC program.
320-07D2	**Error message** PLC: more than one strobe active **Cause of error** PLC runtime error: More than one of the functions "PLC positioning," "datum shift," or "spindle orientation" has been activated. **Error correction** Edit the PLC program.
320-07D3	**Error message** PLC: undefined run-time error **Cause of error** Undefined run-time error **Error correction** Inform your service agency.
320-07D4	**Error message** Supply voltage missing at X44 **Cause of error** No supply voltage is applied at connector X44 of the control (= Relay external DC voltage missing). Possible causes: - PLC fuses defective - 24V power supply unit in the electrical cabinet defective - Faulting wiring of 24 V supply - Lines interrupted **Error correction** - Check the PLC fuses - Check the 24 V power supply unit in the electrical cabinet - Check the wiring of the 24 V supply - Check the lines for interruption
320-07D5	**Error message** Incorrect wiring of EMERGENCY STOP **Cause of error** - The wiring of the emergency stop circuit is faulty. **Error correction** - Inform your service agency - Check the wiring

Error number	Description
320-07D6	**Error message** STOP through PLC **Cause of error** The system was stopped by the PLC program **Error correction** Note any further messages. If necessary, inform your machine tool builder. The control must be shut down and restarted.
320-07D7	**Error message** M0-999 and B0-127 deleted **Cause of error** You called the function for deleting the non-volatile data. The system has been stopped. **Error correction** Restart the control
320-07D8	**Error message** Exchange buffer battery **Cause of error** The voltage of the buffer battery has dropped below the minimum permissible value. **Error correction** Refer to your machine manual or contact your machine tool builder.
320-07D9	**Error message** Temperature too high (CPU%1 := %2°C) Temperature of MC main computer too high := %1°C Temperature of the MC main computer too high: %1°C (warning) **Cause of error** Temperature sensor detects an excessively high temperature within the main computer's housing. - Insufficient heat dissipation for the MC main computer - Contaminated filter pads - Defective climate control unit in the electrical cabinet or operating panel (where MC is installed) - Defective fan - Defective temperature sensor - Unfavorable mounting of components **Error correction** - Clean the filter pads - Check the climate control unit and repair it if necessary - Replace the fan - Inform your service agency

Error number	Description
320-07DB	**Error message** MC housing fan defective **Cause of error** Rotational speed of the housing fan is too low. **Error correction** - Exchange the fan - inform your service agency
320-0BB8	**Error message** %1 **Cause of error** **Error correction**
320-0FA1	**Error message** M **Cause of error** **Error correction**
320-0FA2	**Error message** S **Cause of error** **Error correction**
320-0FA3	**Error message** T0 **Cause of error** **Error correction**
320-0FA4	**Error message** TOOL CALL **Cause of error** **Error correction**
320-0FA5	**Error message** TOOL DEF **Cause of error** **Error correction**
320-0FA6	**Error message** Strobe **Cause of error** **Error correction**

Error number	Description
320-0FA7	**Error message** Acknowledgment **Cause of error** **Error correction**
320-0FA8	**Error message** Parameter **Cause of error** **Error correction**
320-0FA9	**Error message** Rotational speed **Cause of error** **Error correction**
320-0FAA	**Error message** Mode **Cause of error** **Error correction**
320-0FAB	**Error message** Gear range **Cause of error** **Error correction**
320-0FAC	**Error message** Index **Cause of error** **Error correction**
320-0FAD	**Error message** Magazine **Cause of error** **Error correction**
320-0FAE	**Error message** Pocket **Cause of error** **Error correction**

Error number	Description
320-0FAF	**Error message** Channel **Cause of error** **Error correction**
320-0FB0	**Error message** Status **Cause of error** **Error correction**
320-0FB1	**Error message** Manual **Cause of error** **Error correction**
320-0FB2	**Error message** Incremental **Cause of error** **Error correction**
320-0FB3	**Error message** Referencing **Cause of error** **Error correction**
320-0FB4	**Error message** Probing **Cause of error** **Error correction**
320-0FB5	**Error message** Negative direction **Cause of error** **Error correction**
320-0FB6	**Error message** Position **Cause of error** **Error correction**

Error number	Description
320-0FB7	**Error message** Feed rate **Cause of error** **Error correction**
320-0FB8	**Error message** W/o limit switches **Cause of error** **Error correction**
320-0FB9	**Error message** Without NC stop **Cause of error** **Error correction**
320-0FBB	**Error message** Supply voltage missing for HSCI device **Cause of error** It was detected that the +24 V supply voltage was missing on an HSCI bus device. The -PF.BOARD is pending in the S status. **Error correction** - Open the bus diagnosis and check the status messages of the devices connected to the HSCI bus system. - Provide all devices connected to the HSCI bus system with the correct +24 V NC and +24 V PLC voltage. - Restart the control.
320-0FBC	**Error message** The TNC will be shut down and machine functions will be performed **Cause of error** The PLC program is delaying shutdown of the control **Error correction** Run the machine functions requested by the PLC program.
320-0FBF	**Error message** More than one error class was defined for a PET error **Cause of error** - More than one error class was defined for an error in an error table (.PET). **Error correction** - Define only one error class for the relevant error. - You will find more informationen (table name, error number) in the error list through the "Internal info" soft key. - Inform your service agency.

Error number	Description
320-0FC0	**Error message** Fast PLC input has incorrect parameters **Cause of error** A fast PLC input is not in the IOC file, or the parameters are incorrect. **Error correction** - Check the IO configuration. - Inform your service agency
320-0FC2	**Error message** JHIOSim installation incomplete **Cause of error** Windows: JHIOSim DLL not available Virtual Box: Installation incomplete Virtual Box: Common folder /mnt/sf/IOsim does not exist **Error correction** Check the installation of virtual box and Windows
320-0FC3	**Error message** Start of an NC program not permitted at this time **Cause of error** Starting the NC program is not permitted at this time, since axes are currently executing positioning tasks. **Error correction** Try to start the NC program at a later time
320-0FC4	**Error message** System overload – message buffer is full **Cause of error** Very many small jobs were sent in a short time to the PLC run-time system. As a result, the associated messages could no longer be delivered in the system. These jobs result from calling PLC modules or calling the Python function library for OEM applications. The capacity for buffering the messages in the system has been exhausted; the PLC program has been stopped. **Error correction** Change the PLC program or the OEM application Distribute the calls from PLC modules or functions of the JH function library in Python OEM applications more evenly over time.

Error number	Description
320-0FC7	**Error message** Temperature of the MC main computer too low: %1°C (warning) Temperature of the MC main computer too low: %1°C **Cause of error** Temperature sensor detects an excessively low temperature within the main computer's housing. - Defective climate control unit in the electrical cabinet or operating panel (where MC is installed) - Defective temperature sensor - Unfavorable mounting of components **Error correction** - Check the temperature conditions in the electrical cabinet - Check the climate control unit, and repair it if necessary - Inform your service agency
322-0002	**Error message** Incorrect tool location at start of block scan (%1) **Cause of error** Mid-program startup started with incorrect tool. **Error correction** Change to the correct tool and start again.
322-0003	**Error message** Configuration datum %1 - %2 contains errors **Cause of error** The file entered in the configuration data for the description of a tool-change sequence contains errors, and was not loaded onto the control for operation. **Error correction** Inform your machine tool builder.
322-0004	**Error message** Tool call unclear **Cause of error** Two tools are to be inserted or exchanged at the same time. **Error correction** Correct the NC program.
322-0005	**Error message** File not found **Cause of error** The file entered in the configuration data for describing a tool-change sequence does not exist. **Error correction** Inform your machine tool builder.

Error number	Description
322-0006	**Error message** Pocket table faulty **Cause of error** No primary key (TOOL_P.P, magazine number and pocket number) is defined for the pocket table TOOL_P. **Error correction** Inform your machine tool builder.
322-0007	**Error message** Pocket table faulty **Cause of error** The wrong column (not TOOL_P.P, magazine number and pocket number) was defined as primary key for the pocket table TOOL_P. **Error correction** Inform your machine tool builder.
322-0008	**Error message** Pocket table faulty **Cause of error** An invalid value was read in column P of the pocket table. **Error correction** Inform your machine tool builder.
322-0009	**Error message** Pocket table faulty **Cause of error** An invalid value was read in column ST of the pocket table. **Error correction** Inform your machine tool builder.
322-000A	**Error message** Pocket table faulty **Cause of error** An invalid value was read in column F of the pocket table. **Error correction** Inform your machine tool builder.
322-000B	**Error message** Pocket table faulty **Cause of error** An invalid value was read in column PTYP of the pocket table. **Error correction** Inform your machine tool builder.

Error number	Description
322-000C	**Error message** Pocket table faulty **Cause of error** A tool was found more than once in the pocket table. **Error correction** Correct the pocket table. Delete the tool from the pockets that it is not in, or mark these pockets as reserved.
322-000D	**Error message** Pocket table might be inconsistent **Cause of error** The internal data of the control on the loaded tool do not correspond with the content of the pocket table. This can happen after the pocket table is edited or overwritten, or after a tool change was canceled. The contents of the pocket table might no longer correspond to the tool configuration in the tool memory. **Error correction** Check the contents of the pocket table and correct them if required.
322-000E	**Error message** Pocket table faulty **Cause of error** A required column (P, Tor RSV) is not defined for the pocket table TOOL_P. **Error correction** Inform your machine tool builder.
322-000F	**Error message** File not found **Cause of error** The file with the pocket table was not found in the path configured. **Error correction** Restore the file to the original path, or create a new file with the original path, or contact your machine tool builder.
322-0011	**Error message** Access to pocket table denied **Cause of error** The tool call cannot proceed because write-access to the pocket table was denied. The PLC program or editing of the pocket table can be the cause of such a lock. **Error correction** Conclude editing of the pocket table, or contact your machine tool builder.

Error number	Description
322-0012	**Error message** Pocket table faulty **Cause of error** The configuration data for a column in the pocket table are missing or faulty. **Error correction** Inform your machine tool builder
322-0013	**Error message** File access not possible **Cause of error** The file with the pocket table is missing, or access to this file was denied. **Error correction** Restore the file, or check the access rights to the file. Rescind any active write-protection.
322-0014	**Error message** Pocket table faulty **Cause of error** The file with the pocket table is incomplete, or contains lines of differing lengths, syntactical errors or unknown columns. **Error correction** Correct the file, or restore the file, or contact your machine tool builder.
322-0015	**Error message** Pocket table faulty **Cause of error** An invalid value was read in column T of the pocket table. **Error correction** Correct the file, or restore the file, or contact your machine tool builder.
322-0016	**Error message** System error in the PLC **Cause of error** Internal software error **Error correction** Inform your service agency

Error number	Description
322-0017	**Error message** Access to pocket table denied **Cause of error** Access to the pocket table is locked by another application. The control could therefore no longer update the pocket table after tool change. The contents of the pocket table might no longer correspond to the tool configuration in the tool memory. **Error correction** Check the contents of the pocket table and correct them if required. Conclude editing of the pocket table, or contact your machine tool builder.
322-0018	**Error message** System error in the PLC **Cause of error** Internal software error **Error correction** Inform your service agency
322-0019	**Error message** Pocket table faulty **Cause of error** An invalid value was read in column RSV of the pocket table. **Error correction** Correct the file, or restore the file, or contact your machine tool builder.
322-001A	**Error message** Tool change without configured spindle required **Cause of error** No spindle was configured for channel %1 or (only channel 0). It was not possible to implicitly assign a spindle. **Error correction** Edit the NC program or contact your machine tool builder

Error number	Description
322-001B	**Error message** PLC datum %1 cannot be changed **Cause of error** An attempt was made to change the PLC datum with the given designation. The datum does not exist, or cannot be changed, or the new value is not permissible. **Error correction** Check whether the datum with the given designation exists and is free for changes. Check whether the new value is permissible for this datum. Edit your NC program so that a correct designation and a permissible value are used.
322-001C	**Error message** System error in the PLC System error in the PLC **Cause of error** Internal software error **Error correction** Inform your service agency
322-001D	**Error message** Pocket table faulty **Cause of error** An invalid value was read in column L of the pocket table. **Error correction** Inform your machine tool builder
322-001E	**Error message** Pocket in tool magazine is locked **Cause of error** The tool call cannot be executed because the pocket in the tool magazine is locked. Such a lock can be caused by editing the pocket table or by the PLC program. **Error correction** Correct the pocket datum or inform your machine manufacturer.
330-0002	**Error message** Safe Torque Off (-STO.B.x) is active **Cause of error** - Error in program run - Safety function Safe Torque Off (STO) is active **Error correction** - Inform your service agency.

Error number	Description
330-0004	**Error message** Switch off external dc voltage **Cause of error** The machine control voltage is still switched on. **Error correction** Switch off the machine control voltage.
330-0005	**Error message** Normally closed relay open? **Cause of error** In the relay chain the normally closed contact of one or more relays is open. **Error correction** - Check the relay for proper function. - If necessary, contact your service agency.
330-0006	**Error message** Switch on external dc voltage **Cause of error** The machine control voltage is switched off. **Error correction** Switch off the machine control voltage.
330-0008	**Error message** Inverter not ready **Cause of error** - Power-on of the drive not possible, because an inverter is not ready (RDY signal). - On interface PCBs for Siemens inverters, the second axis is not enabled - No switch signals to the inverter contacts or relays - Compact inverter, inverter supply unit or power module is defective - Interruption at inverter bus cable (supply bus, unit bus, PWM bus) - Defective PWM interface on the control **Error correction** - Remove interruption in the electrical cabinet - Replace the defective compact inverter, supply unit or power module - Replace the defective cable - Inform your service agency

Error number	Description
330-0012	**Error message** MC machine key depressed **Cause of error** Contact of a machine key does not open. **Error correction** Release the key if pressed, otherwise inform your service agency.
330-0013	**Error message** Relay: n.c. contact closed? **Cause of error** In the relay chain, the normally closed contact of all relays is closed. **Error correction** Check the relays for proper function. If required, inform your service agency.
330-0014	**Error message** CC%2 inverter for spindle RDY=0 **Cause of error** The power module of the spindle could not be switched to ready condition. - Safety relay not on (e.g. connector X71 of the UE/UV/UVR, X73 of the HEIDENHAIN expansion board for Simodrive) - Defective power module - PWM bus cable interrupted **Error correction** Check the wiring and inform your service agency.
330-0015	**Error message** CC%2 inverter for axis RDY=0 **Cause of error** The power module of an axis could not be switched to ready condition. - Safety relay not on (e.g. connector X72 of the UV, X73 of the HEIDENHAIN expansion board for Simodrive) - Defective power module - PWM bus cable interrupted **Error correction** Check the wiring and inform your service agency.
330-0016	**Error message** CC%2 inverter for spindle RDY=1 **Cause of error** The spindle power supply is ready for operation although it ought to be switched off. **Error correction** Inform your service agency.

Error number	Description
330-0017	**Error message** CC%2 inverter for axis RDY=1 **Cause of error** The power module of an axis is ready for operation although it should actually be switched off. **Error correction** Inform your service agency.
330-001A	**Error message** Input (ES.B) not equal to 0 **Cause of error** 24 V at input ES.B. During the dynamic test, the voltage is expected to be 0 V. **Error correction** Inform your service agency.
330-001B	**Error message** Test of cutout channels inactive **Cause of error** The MC (Main Computer Unit) failed to test the cutoff channels. **Error correction** Inform your service agency.
330-0020	**Error message** Command buffer overflow **Cause of error** The CC could not run so many commands from the MC. **Error correction** Inform your service agency.
330-0021	**Error message** Commands do not agree **Cause of error** The command confirmed from the CC as an echo is not the command sent from the MC. **Error correction** Inform your service agency.
330-0022	**Error message** CC%2 command not acknowledged **Cause of error** Command was not acknowledged by the Computer Control Unit (CC) within 200 ms. **Error correction** Inform your service agency.

Error number	Description
330-0023	**Error message** FS function not performed **Cause of error** One or more FS functions within a cycle were not performed. **Error correction** Inform your service agency.
330-0024	**Error message** MC handwheel permissive key pressed **Cause of error** The permissive button of the handwheel was pressed. An incorrect handwheel was selected by CfgHandwheel–>type. **Error correction** - Check the permissive button - Set the configuration datum properly; - Inform your service agency.
330-0025	**Error message** Faulty data from CC%2 **Cause of error** Faulty software **Error correction** Inform your service agency.
330-0027	**Error message** MC pos. deviation too large %2 **Cause of error** The calculated position deviation between the speed encoder and the position encoder is greater than the value from the parameter CfgAxisSafety->positionDiffRef or CfgAxisSafety->positionDiffRun. - Excessive difference between the positions calculated from the position encoder pulses and the speed encoder pulses - During initial operation: Incorrect standardization of speed encoder pulses (e.g. incorrect screw pitch entered) - Excessive backlash - Defective coupling, gear, etc. - Belt torn **Error correction** - Switch the control off and on again - Check CfgAxisSafety->positionDiffRef, CfgAxisSafety->positionDiffRun - During initial operation: Check the standardization of speed encoder pulses (enter the correct screw pitch) - Check the backlash - Repair the defective coupling, gear, etc. - Replace the belt - Inform your service agency

Error number	Description
330-0028	**Error message** No pos. values from the CC%2 **Cause of error** For a certain time the CC has not sent any position values to the MC. **Error correction** - Switch the control off and back on again - Inform your service agency
330-0029	**Error message** CC%2 no pos. values from the MC **Cause of error** - The MC must not send any position values to the CC. - Internal software error **Error correction** - Switch the control off and on again - Inform your service agency
330-002A	**Error message** MC/CC%2 checked axes unequal **Cause of error** - Contradictory status of checked position values in the MC and CC. - Internal software error **Error correction** - Switch the control off and on again - Inform your service agency
330-002B	**Error message** CC%2 wrong include-file version **Cause of error** - The version number of one of the called Include files are different in the MC and CC - Internal software error **Error correction** - Check the software version - Inform your service agency
330-002D	**Error message** Switch on spindle **Cause of error** The spindle start key was pressed but not the permissive key while the protective door A/S was open. **Error correction** Press the spindle start key and the permissive key

Error number	Description
330-002E	**Error message** SMP or checksum erroneous **Cause of error** - Safety-related machine parameter (SMP) was changed. - Checksum over safety-related machine parameter was changed. **Error correction** - Check the safety-related machine parameter and the checksum. - Change must be conducted only by the machine manufacturer using a manufacturer's password. - If there are changes, it may be necessary to conduct acceptance tests on the machine. - Inform your service agency
330-002F	**Error message** SMP or checksum calculation erroneous **Cause of error** - Safety-related machine parameter (SMP) was changed. - Checksum over safety-related machine parameter was changed. - Machine parameter file could not be opened or does not exist. **Error correction** - Check the safety-related machine parameter and the checksum. - Change must be conducted only by the machine manufacturer using a manufacturer's password. - If there are changes, it may be necessary to conduct acceptance tests on the machine. - Inform your service agency
330-0030	**Error message** Safe axes must have sinusoidal inputs **Cause of error** - Safety orientation (SG) error **Error correction**
330-0031	**Error message** Axis status cannot be determined **Cause of error** - Safety orientation (SG) error **Error correction**

Error number	Description
330-0033	**Error message** Not in the REF operating mode **Cause of error** - The axes can be tested only in the Reference Run REF operating mode. **Error correction** - Press CE to acknowledge the error message and switch to the REF mode. - Then home the axes. - Inform your service agency
330-0034	**Error message** Invalid axis **Cause of error** - The axis to be tested is not a safe axis. - The axis to be tested is a spindle. - The given axis number is invalid. **Error correction** - Check the configuration of the axis and, if necessary, correct it (spindles cannot be referenced). - Inform your service agency.
330-0035	**Error message** Axis has already been checked **Cause of error** - Axis to be tested has already been checked. - The given axis number is invalid. **Error correction** - Internal software error. - Inform your service agency
330-0036	**Error message** Axis is not referenced **Cause of error** - The axis to be tested has not yet been homed. **Error correction** - Internal software error. - Inform your service agency
330-0037	**Error message** Axis in motion **Cause of error** - The axis to be tested is not yet stationary. **Error correction** - Press CE to acknowledge the error message and bring the axis to a standstill. - Then test the axis. - Inform your service agency

Error number	Description
330-0038	**Error message** Axis not at test position **Cause of error** - The axis to be tested is not located at the test position (safe machine parameter positionMatch in CfgAxisSafety). - The axis to be tested is not located at the test position (safe machine parameter positionDiffRef in CfgAxisSafety). - Axis traverse direction is not correctly configured (machine parameter MP_signCorrActualVal, MP_signCorrNominalVal or entry in the DIR column of the motor table). **Error correction** - Press CE to acknowledge the error message and move the axis to the test position. - Then test the axis. - If the message appears although the axis is at the correct test position, check the configuration of the axis traverse direction and correct it if necessary. - Inform your service agency
330-0039	**Error message** Missing permission **Cause of error** - When testing the axis (message: "Confirm with permissive key") control receives no permission through a permissive key. - Permissive key(s) defective. - Test of the axes cannot be concluded without permission. **Error correction** - Check the hardware of the permissive keys. - Inform your service agency
330-003A	**Error message** No authorization for testing **Cause of error** - The axis(or axes) cannot be tested in the safety-related operating mode SOM_1. **Error correction** - Select another safety-related operating mode (e.g. SOM_2, SOM_3). - To acknowledge any pending error message, use the CE key. - Then test the axis. - Inform your service agency if the error message recurs.

Error number	Description
330-003C	**Error message** MC S input signals %2 not equal **Cause of error** Safety-oriented input of SPLC-MC is not equal to input of SPLC-CC. e.g. FB_NCC.x, KSW.x, ES.x **Error correction** Inform your service agency.
330-003E	**Error message** MC limit switch %2 + **Cause of error** - Violation of the absolute positive limit position value (in the positive traverse direction) of the safety function SLP. - The calculated path of the tool exceeds the defined traversing range (software limit switch) of the machine. - The software limit switch (absolute position limit value) was reached in a manual operating mode. **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point. - Move the tool in the opposite direction. - Inform your service agency.
330-0040	**Error message** MC limit switch %2 - **Cause of error** - Violation of the absolute negative limit position value (in the negative traverse direction) of the safety function SLP. - The calculated path of the tool exceeds the defined traversing range (software limit switch) of the machine. - The software limit switch (absolute position limit value) was reached in a manual operating mode. **Error correction** - Check the programmed coordinates. If required, edit the program. - Check the reference point. If required, set a new reference point. - Move the tool in the opposite direction. - Inform your service agency.

Error number	Description
330-0042	**Error message** MC standstill monitoring %2 **Cause of error** - The position error in an active safe operating stop (SOS) is greater than the value specified in the configuration parameter CfgAxisSafety->positionRangeVmin. - If during an emergency stop the axes and spindle are decelerated, and the axes come to a stop earlier than the spindle, then the axes are standstill-monitored for the value from CfgAxisSafety->positionRangeVmin until the spindle comes to a stop. - If the protective door is closed in the automatic mode (SOM 1), axes whose drives are switched off (e g. clamped axes) are standstill-monitored for the value from CfgAxisSafety->positionRangeVmin. - After the permissive button is released, the tool magazine axis continues to move for more than 3 seconds. - During the braking test, the axis exceeds the maximum permissible path in CfgAxisSafety->positionRangeVmin. It must be assumed that a holding brake for the axis is defective! - After triggering of an SS2 stop reaction and expiration of the SS2 stopping time, a position error greater than the value in the configuration parameter CfgAxisSafety->positionRangeVmin was detected. **Error correction** - If the error occurs during the brake test, the axis remains in the servo loop. The axis has to be moved to a safe position before you switch the machine off. It must be assumed that a holding brake is defective. - Inform your service agency.
330-0044	**Error message** MC feed rate greater SLS %2 **Cause of error** The feed rate exceeds the permissible limit value for the active safety-related operating mode (SOM). **Error correction** Inform your service agency.
330-0045	**Error message** CC feed rate greater SLS %2 **Cause of error** The feed rate exceeds the permissible limit value for the active safety-related operating mode (SOM). **Error correction** Inform your service agency.

Error number	Description
330-0046	**Error message** MC S input %2 not equal 0 CC%1 S input %2 not equal 0 **Cause of error** The safety-oriented inputs for detachable-key switches, door contacts and emergency stop were not set to 0 during the cyclic test. **Error correction** Inform your service agency!
330-0048	**Error message** MC NC temperature out of tol. CC%2 NC temperature out of tol. **Cause of error** The temperature inside the control is outside the permissible tolerance range. - Clogged filter pads in the electrical cabinet - Defective climate control unit in the electrical cabinet - Defective fan in the control housing - Defective control **Error correction** - Clean the filter pads - Repair the air conditioner - Exchange the fan in the control housing / if necessary, exchange the complete control Improve the ventilation in the electrical cabinet.
330-004A	**Error message** MC +5V out of tolerance **Cause of error** - The internal +5-V power supply of the MC is outside the permissible tolerance range. **Error correction** - Exchange defective hardware (MC) - Inform your service agency
330-004B	**Error message** CC%2 +5V out of tolerance **Cause of error** The 5V power supply of the control is outside the permissible tolerance range. **Error correction** Inform your service agency

Error number	Description
330-004C	**Error message** Op. state of MC not equal CC **Cause of error** The automatic SLS, SOS, STO operating states of the MC and CC are compared cyclically. If the values remain unequal for longer than 500 ms, a Safe Stop 1 (SS1) is released. **Error correction** - Press CE to acknowledge the error message - Switch the machine off and on - Inform your service agency. - Check the software version
330-004D	**Error message** CC%2 operating state not equal MC **Cause of error** - The automatic SLS, SOS, STO operating states of the MC and CC are compared cyclically. If the values remain unequal for longer than 500 ms, a Safe Stop 1 (SS1) is released. **Error correction** - Press CE to acknowledge the error message - Switch the machine off and on - Inform your service agency. - Check the software version
330-004E	**Error message** MC amplitude too high %2 **Cause of error** The amplitude of the encoder signal is too high or the signal for contamination is active. - Incorrect adjustment between head and encoder, air gap too small (exposed encoders) - Excessive supply voltage **Error correction** - Check the amplitude of the encoder signal - Inform your service agency

Error number	Description
330-0050	**Error message** MC amplitude too low %2 CC amplitude too low %2 Disturbance of handwheel for %2 (handwheel on encoder connection) **Cause of error** The amplitude of the encoder signal is too small or the signal for contamination is active. - Encoder contaminated - Encoder defective - Penetration of humidity - Scanning head misaligned (distance, parallelism, etc.) - Encoder cabling defective - Encoder input defective on the control - Vibration - Interfering signals **Error correction** - Inform your service agency
330-0052	**Error message** MC frequency too high %2 **Cause of error** The maximum input frequency was exceeded at an encoder input. - Fault on signal of the speed encoder - Vibrations on the machine **Error correction** - Check the speed encoder connection (ground connection) - Check the speed encoder - Check the encoder signal input frequency. - Eliminate the vibrations - Inform your service agency
330-0054	**Error message** MC speed greater than SLS %2 **Cause of error** The rotational speed exceeds the permissible limit value for the active safety-related operating mode (SOM_2, SOM_3, SOM_4 or SLI_S). **Error correction** Inform your service agency.
330-0055	**Error message** CC speed greater than SLS %2 **Cause of error** The feed rate exceeds the permissible limit value for the active safety-related operating mode (SOM). **Error correction** Inform your service agency.

Error number	Description
330-0056	**Error message** MC: Machine permissive button depressed **Cause of error** The permissive button of the tool changer was pressed. **Error correction** - Check the permissive buttons. - Contact your service agency.
330-0057	**Error message** CC%2 SH1=0 MC -SMC.A.WD=0 **Cause of error** Internal software error **Error correction** Inform your service agency
330-0058	**Error message** CC%2 SH1=1 **Cause of error** - Hardware error - Internal software error **Error correction** Inform your service agency
330-0059	**Error message** No hardware for functional safety **Cause of error** Hardware components not compatible with safety software. **Error correction** Inform your service agency.
330-005A	**Error message** MC no memory available **Cause of error** Internal software error **Error correction** Inform your service agency

Error number	Description
330-005B	**Error message** MC safety doors are open **Cause of error** Guard doors are open The guard doors have to be closed in order to conduct the safety self-test or the brake test of an axis. Further possible causes: - Faulty wiring of guard doors - Guard door contact is defective - SPLC interface signal PP_AxGrpStateReq unequal to S_S-TATE_AUTO [10] **Error correction** - Close the guard doors in order to conduct the safety self-test or the brake test Further possible measures: - Check the wiring of the guard door contacts - Check the guard door contact. - Check the SPLC program. - Inform your service agency
330-005C	**Error message** Axis %2 started w/o test by MC **Cause of error** The axis was not yet tested. **Error correction** Test the axis.
330-005E	**Error message** Internal software error Internal software error **Cause of error** Internal software error **Error correction** Inform your service agency.
330-005F	**Error message** FS hardware used without FS software! **Cause of error** The safety hardware cannot be installed without safety software! **Error correction** Inform your service agency.

Error number	Description
330-0060	**Error message** FS error in configuration **Cause of error** - The machine parameter or config object is erroneous, incomplete, or the like (datum was read as additional info). - Axis group from config object CfgAxisSafety is not configured. - Safe spindle not in the axis group for spindles - Maximum velocity for SOM 2 is missing or incorrect - Safe feed axis not in the axis group for spindles - The encoder input (position or motor) was configured incorrectly or not at all - PWM output was configured incorrectly - Safe axis not controlled through CC (analog, simulation, PLC spindle, etc.) - Safe axis configured for “referencing on the fly"; check machine parameter MP_refType (400401) - Differing line counts for motor encoder in various parameter sets of one axis - Several axis groups assigned with identical ID - Invalid index number assigned for axis group (negative, too large) **Error correction** - Correct the machine configuration - Inform your service agency
330-0064	**Error message** MC brake test not completed **Cause of error** The brake test was started, but no end was received within 10 seconds from the CC. **Error correction** - Internal software error - Inform your service agency
330-0065	**Error message** MC command %1 not acknowledged **Cause of error** Command was not acknowledged by the Main Computing Unit (MC) within 400 ms. **Error correction** Inform your service agency.
330-0066	**Error message** MC PWM output is not present **Cause of error** Internal software error **Error correction** Inform your service agency

Error number	Description
330-0067	**Error message** CC%2 timeout version comparison **Cause of error** The CC does not send a version number (safe stop 0) within 2 s. **Error correction** Inform your service agency.
330-0068	**Error message** MC wrong quantity FS-CC **Cause of error** Internal software error **Error correction** - Inform your service agency - Exchange the software
330-0069	**Error message** MC S checksum error CC%2 FS checksum error **Cause of error** - Checksum error due to faulty data - Internal software error **Error correction** Inform your service agency.
330-006A	**Error message** CC%2 timeout SMP check sum **Cause of error** The CC does not find an SMP checksum (safe stop 0) within 2 s. - Internal software error **Error correction** Inform your service agency.
330-006B	**Error message** MC SOM 2 only one axis allowed **Cause of error** Two or more axes are to be moved simultaneously in the SOM 2 operating mode. **Error correction** Only one axis may be moved at a time in SOM 2.

Error number	Description
330-006C	**Error message** +24 V MC short on brake line **Cause of error** There is a short circuit with 24 V on the brake line A of the MC. **Error correction** - Check brake channel A - Inform your service agency
330-006D	**Error message** MC 0 V short on brake line %2 **Cause of error** There is a short circuit with 0 V on the brake line A of the MC. **Error correction** - Check brake channel A - Inform your service agency
330-006E	**Error message** MC SOM 4 Press the permissive key **Cause of error** In the SOM 4 mode of operation, the permissive key was not pressed within the time defined in MP529. **Error correction** Press the permissive key.
330-006F	**Error message** MC SOM 4 not released **Cause of error** The operating mode BA4 is addressed through the keylock switch and is still not enabled over the configuration datum "permitSom4". **Error correction** - Inform your service agency - Enable the operating mode through the configuration datum "permitSom4"
330-0070	**Error message** MC SOM 4 not possible **Cause of error** 1. Key switch is not on SOM 1. **Error correction** 1. Turn the key switch to SOM 1.

Error number	Description
330-0071	**Error message** MC operating mode not possible **Cause of error** Wiring error at the SG inputs BA2.x, BA3.x and BA4.x. **Error correction** - Inform your service agency. - Check the wiring.
330-0073	**Error message** CC%2 current measurement timeout **Cause of error** Current measurement of the controller unit in the cut-out test could not be performed because the standby signal RDY of a power stage (inverter) is missing. **Error correction** - Check the switch setting of "AXIS/SPINDLE (X110)" on the inverter. - Inform your service agency
330-0074	**Error message** MC: Incorrect safe axis group **Cause of error** - The axis-group definition is not allowed in the safe machine parameter axisGroup in CfgAxisSafety. **Error correction** - Correct the machine parameter axisGroup in CfgAxisSafety. - Changes in this safe machine parameter may be performed solely by the machine tool builder. - Inform your service agency.
330-0075	**Error message** MC braking test not activated **Cause of error** - Move the axis to a safe position before power-off - Brake test was not commanded by the MC **Error correction** - Software error - Inform your service agency

Error number	Description
330-0076	**Error message** CC: Guard doors are open **Cause of error** - One or more guide doors are open The guard doors have to be closed in order to conduct the safety self-test or the brake test of an axis. Further possible causes: - Faulty wiring of guard doors - Guard door contact is defective **Error correction** - Close the guard doors in order to conduct the safety self-test or the brake test Further possible measures: - Check the wiring of the guard door contacts - Check the guard door contact - Check the SPLC program - Inform your service agency
330-0077	**Error message** Checksum error A **Cause of error** The CRC sum of the EPROMs IC-P1 and IC-P2 is incorrect. **Error correction** - Switch the control off and on again (reboot) - Inform your service agency.
330-0078	**Error message** MC movement monitoring %2 **Cause of error** - Defective hardware - Spindle cannot be controlled **Error correction** - Adjust the spindle - Exchange the hardware - Inform your service agency
330-0079	**Error message** MC error in braking process %2 **Cause of error** The axes cannot be servo-controlled. dv/dt monitoring has responded. **Error correction** - Check machine parameter CfgAxParSafety/timeTolerанceDvDt - Adjust the axes - Exchange the hardware - Inform your service agency

Error number	Description
330-007A	**Error message** CC: Machine key depressed **Cause of error** One or more machine keys of an operating station are active on the CC side. Machine keys are keys, e.g. on the machine operating panel or the handwheel, that can release a movement of the machine (permissive keys, axis-direction keys, chip conveyors, etc.) **Error correction** - Release the key(s) or check the wiring - Inform your service agency
330-007B	**Error message** MC: Mach. permiss. button depressed **Cause of error** Machine-key permissive key is depressed on the MC side. **Error correction** - Release the key or check the wiring - Inform your service agency
330-007C	**Error message** CC panel permissive button pressed **Cause of error** Machine-key permissive key is depressed on the CC side. **Error correction** - Release the key or check the wiring - Inform your service agency
330-007D	**Error message** CC: Machine permissive button pressed **Cause of error** Tool changer permissive key is active on CC side. **Error correction** - Release the key or check the wiring - Inform your service agency
330-007E	**Error message** MC inverter RDY=0 %2 **Cause of error** Although SH1B.x=1, RDY.x=0 **Error correction** - Hardware error - Inform your service agency

Error number	Description
330-007F	**Error message** MC inverter RDY=1 %2 **Cause of error** Although SH1B.x=0, RDY.x=1 **Error correction** - Hardware error - Inform your service agency
330-0080	**Error message** MC 1st violation of limit switch %2+ **Cause of error** The safety limit switch was tripped for the first time. **Error correction** - Press "machine on" to cancel the error message and move to the limit switch range.
330-0081	**Error message** MC 1st violation of limit switch %2- **Cause of error** The safety limit switch was tripped for the first time. **Error correction** - Press "machine on" to cancel the error message and move to the limit switch range.
330-0082	**Error message** MC nominal-to-actual position error too large %2 **Cause of error** The following error of a moving axis is greater than the value specified in the safe machine parameter positionDiffNom. **Error correction** - Reduce the contouring feed rate, increase the rotational speed. - Remove any possible sources of vibration. - Inform your service agency if the error occurs frequently.
330-0083	**Error message** MC S software error %1 **Cause of error** - Software error **Error correction** - Inform your service agency

Error number	Description
330-0084	**Error message** MC start test of the cut-out channels not possible **Cause of error** The start test of the cutout channels was commanded through the PLC, although the machine-on input is inactive. **Error correction** - Error in PLC program - Inform your service agency
330-0085	**Error message** CC: Handwheel permissive button depressed **Cause of error** The permissive button of the handwheel on the CC side was pressed. An incorrect handwheel was selected by CfgHandwheel->type. **Error correction** - Check the permissive button - Set the configuration datum properly - Inform your service agency
330-0086	**Error message** Emergency stop from SPLC **Cause of error** - Stop reaction for emergency stop (SS1) released by the SPLC program **Error correction** - Inform your service agency
330-0087	**Error message** Fatal error from SPLC **Cause of error** - Stop reaction for fatal error (SS1) released by the SPLC program **Error correction** - Inform your service agency
330-0088	**Error message** MC: Permissive button depressed **Cause of error** Permissive button of machine operating panel or handwheel is depressed. **Error correction** - Release permissive button - Check inputs

Error number	Description
330-0089	**Error message** CC permissive button pressed **Cause of error** Permissive button of machine operating panel or handwheel is depressed. **Error correction** - Release permissive button - Check inputs
330-008A	**Error message** CC%2 inverter for axis RDY=0 (safety relay) **Cause of error** RDY of the inverter remains 0, although 1 is expected after a beginning a test. Possible causes: - Safety relay for axes (K1) has a short circuit to 0 V - Wiring of the enabling relays (X72) or the axis release module is faulty - Wiring of the STO.A.G or MC.RDY signal is faulty - Inverter is not configured for axes **Error correction** - Check the hardware - Check the wiring - Inform your service agency
330-008B	**Error message** CC%2 inverter for axes RDY=1 (safety relay) **Cause of error** RDY of the inverter remains 1, although 0 is expected after a beginning a test. Possible causes: - Safety relay for axes (K1) has a short circuit to +24 V. - Wiring of the enabling relays (X72) or the axis release module is faulty - Wiring of the STO.A.G or MC.RDY signal is faulty - Inverter is not configured for axes **Error correction** - Check the hardware - Check the wiring - Inform your service agency
330-008C	**Error message** MC timeout during braking (SS1) %2 **Cause of error** - The maximum permissible time for braking at the current limit (SS1 reaction) was exceeded. **Error correction** - Check the parameter values: timeLimitStop1: Default time for stopping the axes/spindle at the emergency braking ramp for SS1 reaction - Inform your service agency.

Error number	Description
330-008D	**Error message** MC traverse range exceeded at SS2 %2 **Cause of error** When braking at a contour (SS2), a spindle or axis exceeded the maximum permissible path. The maximum permissible path for axes is the entry in the safety-related machine parameter distLimitStop2. A maximum of 2 revolutions are permissible for spindles during an SS2 reaction in SOM_2, and 5 revolutions are permissible in SOM_3, SOM_4. **Error correction** - Check the machine parameters for the braking process during an SS2 reaction. - Check the entry in distLimitStop2 - Inform your service agency.
330-008E	**Error message** MC change from SOM_2/SOM_3 to SOM_4 not possible MC change from SOM_4 to SOM_2/SOM_3 not possible **Cause of error** - Safety-related mode of operation SOM_1 not selected - E.g. keylock switch 1 not in SOM_1 position **Error correction** - Select safety-related mode of operation SOM_1 - E.g. keylock switch 1 in SOM_1 position
330-0090	**Error message** MC SPLC requests invalid stop reaction **Cause of error** - SPLC demands an invalid stop reaction (SS0, SS1, SS1F or SS2) for an axis/spindle group - Error in SPLC program **Error correction** - Check the SPLC program - Inform your service agency
330-0091	**Error message** MC SPLC requests invalid safety function **Cause of error** - SPLC demands an invalid safety function (SLI, SLS, STO, SOS or AUTO) for an axis/spindle group - Error in SPLC program **Error correction** - Check the SPLC program - Inform your service agency

Error number	Description
330-0092	**Error message** MC brake cannot be deactivated %2 **Cause of error** - Drive was switched on although the STO (Safe Torque Off) safety function for the axis or spindle is still active. **Error correction** - Internal software error - Inform your service agency
330-0093	**Error message** MC System clock MC is not equal to SKERN MC **Cause of error** - MC does not increase the internal watchdog counter value - There is an MC hardware defect or internal software error **Error correction** - Restart the control - Inform your service agency
330-0094	**Error message** MC System clock MC is not equal to SPLC MC **Cause of error** - MC does not get any messages from the SPLC MC - There is an SPLC MC hardware defect or internal software error **Error correction** - Restart the control - Inform your service agency
330-0095	**Error message** MC System clock MC is not equal to SPLC CC **Cause of error** - MC does not get any messages from the SPLC CC - There is an SPLC CC hardware defect or internal software error **Error correction** - Restart the control - Inform your service agency

Error number	Description
330-0096	**Error message** MC error during cross comparison: %1 **Cause of error** - A cross comparison between safety-related data of the MC and the CC reports errors - Data of the programming interface SPlcApiFromSaftey (NN_xxx) are different on the MC and CC - Data of the programming interface SPlcApiToSaftey (PP_xxx) are different on the MC and CC - Hardware defective- Internal software error **Error correction** - Inform your service agency
330-0097	**Error message** MC Safe output is not equal: %1 **Cause of error** Cross comparison of a back-readable output reports an error. Possible causes: - Power supply of the outputs - Wiring of the outputs - Hardware defective (PL module) - Internal software error **Error correction** - Check the power supply of both channels or all outputs - Check the wiring - Exchange the defective PL module - Inform your service agency
330-0098	**Error message** MC +3.3 V out of tolerance **Cause of error** - The internal +3.3-V power supply of the MC is outside the permissible tolerance range. **Error correction** - Exchange defective hardware (MC) - Inform your service agency
330-0099	**Error message** MC +3.3 V PIC out of tolerance **Cause of error** - The internal +3.3 V PIC power supply of the MC is outside the permissible tolerance range. **Error correction** - Exchange defective hardware (MC) - Inform your service agency

Error number	Description
330-009A	**Error message** MC +12 V out of tolerance **Cause of error** - The internal +12-V power supply of the MC is outside the permissible tolerance range. **Error correction** - Exchange defective hardware (MC) - Inform your service agency
330-009B	**Error message** MC speed of fan 1 too slow **Cause of error** - Internal fan 1 of the MC is below the permissible tolerance range (too slow). **Error correction** - Exchange defective hardware (MC) - Inform your service agency
330-009C	**Error message** MC Speed of fan 2 too slow **Cause of error** - Internal fan 2 of the MC is below the permissible tolerance range (too slow). **Error correction** - Exchange defective hardware (MC) - Inform your service agency
330-009D	**Error message** MC fan not recognized **Cause of error** - Hardware (MC) defective **Error correction** - Exchange defective hardware (MC) - Inform your service agency
330-009E	**Error message** MC error in actual-value measurement %2 %1 **Cause of error** The encoder reports an internal error in the actual value acquisition **Error correction** Inform your service agency

Error number	Description
330-00A0	**Error message** MC axis %2 in servo control **Cause of error** - The SPLC program requests the STO safety function, although the axis is still in servo control - Internal software error **Error correction** Inform your service agency
330-00A1	**Error message** MC spindle %2 in servo control **Cause of error** - The SPLC program requests the STO safety function, although the spindle is still in servo control - Internal software error **Error correction** Inform your service agency
330-00A2	**Error message** MC excessive deviation of actual from nominal speed %2 **Cause of error** - The safety function "comparison of actual to nominal speed values" reports an error - The maximum permissible deviation between the actual and nominal speed value (speedDiffNom) exceeded the permissible time in the safety related machine parameter timeToleranceSpeed **Error correction** - Check the entries in the safety-related machine parameters speedDiffNom and timeToleranceSpeed in CfgAxisSafety - Inform your service agency
330-00A3	**Error message** MC: Erroneous data from SPLC %1 **Cause of error** - Data transfer error - CRC checksum error **Error correction** - Inform your service agency.

Error number	Description
330-00A4	**Error message** MC S status reaction is active: %1 **Cause of error** - An error bit was set in the S status by an internal hardware or software error: -STO.B.CC.WD: Watchdog WD.B.CC of a CC controller unit has timed out -SMOP.WD: Watchdog WD.A.SMOP or WD.B.SMOP of an MB machine operating panel has timed out -SPL.WD: Watchdog WD.A.SPL or WD.B.SPL of a PLB has timed out -PF.BOARD: The internal voltage monitoring of the HSCI components has detected a faulty operating voltage -REQ.SS2: The internal temperature or fan monitoring of the HSCI components has detected a fault **Error correction** - Inform your service agency.
330-00A6	**Error message** MC SMC autotest software loaded **Cause of error** - Unsafe test software loaded for acceptance tests - Caution: Safety functions have been partly deactivated! **Error correction** - Inform your service agency.
330-00A8	**Error message** Checksum of safe machine parameters are invalid **Cause of error** No checksum has been saved or a checksum is invalid for at least one of the following configurations: - Safe configuration data - Hardware configuration - Configuration of the encoders **Error correction** - Check the entries of the safe machine parameters and correct them if necessary - If the values of the machine parameters have been changed, they are to be loaded by entering the manufacturer's password. Then a corresponding acceptance test must be run. - Inform your service agency.

Error number	Description
330-00A9	**Error message** Checksum of safe machine parameters have been changed **Cause of error** - The internally saved checksum of the safe machine parameters does not match the new calculated checksum. - One or more safe machine parameters were changed. **Error correction** - Check the entries of the safe machine parameters - If the values of the machine parameters have been changed, they are to be loaded by entering the manufacturer's password. Then a corresponding acceptance test must be run. - Inform your service agency.
330-00AA	**Error message** Checksum was changed through the hardware configuration **Cause of error** - The internally saved checksum of the HSCI system hardware configuration does not match the new calculated checksum. - HSCI components were exchanged, removed, or new ones inserted **Error correction** - Check the hardware configuration and correct it if required. - If the configuration has been changed, they are to be loaded by entering the manufacturer's password. Then a corresponding acceptance test must be run. - Inform your service agency.
330-00AB	**Error message** Checksum was changed through the encoder configuration **Cause of error** - The internally saved checksum of the encoders' configuration does not match the new calculated checksum. - Encoders were exchanged, removed, or new ones inserted **Error correction** - Check the encoder configuration and correct it if required. - If the configuration has been changed, they are to be loaded by entering the manufacturer's password. Then a corresponding acceptance test must be run. - Inform your service agency.

Error number	Description
330-00AC	**Error message** Checksum of safe machine parameters have been changed **Cause of error** Entries in safe machine parameters have been changed. **Error correction** - Check the entries of the safe machine parameters - Switch the control off and on again - If the values of the machine parameters have been changed, they are to be loaded by entering the manufacturer's password. Then a corresponding acceptance test must be run. - Inform your service agency.
330-00AD	**Error message** Impermissible deviation of safe machine parameters Deviations in safe machine parameter **Cause of error** Safe machine parameters deviate between the individual parameter blocks of a safe axis. This is not allowed. The values have to match in all parameter blocks of an axis: - Encoder input - PWM output - Encoder resolution per path - Counting direction - Line count **Error correction** - Check the entries of the safe machine parameters and correct them if necessary - Switch the control off and on again - If the values of the machine parameters have been changed, they are to be loaded by entering the manufacturer's password. Then a corresponding acceptance test must be run. - Inform your service agency.

Error number	Description
330-00AE	**Error message** Invalid entry in safe machine parameter **Cause of error** The configuration datum CfgAxisSafety contains invalid values in one of the following safe machine parameters: - positionMatch Entered position or value is invalid - positionDiffRef Entered deviation or value is invalid - speedLimitSom2 Entered velocity or value is invalid - axisGroup Invalid axis group, too many spindles, too many axes or axis configured as spindle. **Error correction** - Check the entries of the safe machine parameters and correct them if necessary - Switch the control off and on again - If the values of the machine parameters have been changed, they are to be loaded by entering the manufacturer's password. Then a corresponding acceptance test must be run. - Inform your service agency.
330-00AF	**Error message** Invalid entry in safe machine parameter **Cause of error** The configuration datum CfgAxGroupSafety contains an invalid value in a safe machine parameter. - The number of the axis group in the safe machine parameter “id”invalid. - The axis group is not of the “SPINDLE” type, but in the safe machine parameter “brakeAfter” there is a dependency on at least one other axis group. - The axis group is not of the “SPINDLE” type, but the value “STO” is set in the safe machine parameter “idleState”. **Error correction** - Check the entries of the safe machine parameter and correct if necessary - Switch the control off and on again - If the values of the machine parameters have been changed, they are to be loaded by entering the manufacturer’s password. Then a corresponding acceptance test must be run. - Inform your service agency

Error number	Description
330-00B0	**Error message** Invalid entry in safe machine parameter **Cause of error** The safe machine parameter cfgSafety contains invalid values. **Error correction** - Check the entries of the safe machine parameter and correct them if necessary - Switch the control off and on again - If the values of the machine parameters have been changed, they are to be loaded by entering the manufacturer's password. Then a corresponding acceptance test must be run. - Inform your service agency.
330-00B1	**Error message** Invalid entry in safe machine parameter **Cause of error** The configuration of the safe machine parameters is invalid. The following causes are possible: - Invalid encoder input was configured or none and all (speed or position encoder) - Invalid PWM output was configured or none at all - Invalid assignment of axes/spindles to controller main boards - Invalid assignment of PWM output and encoder input (speed or position encoder) to the controller main board - Configuration between PWM output and encoder input (speed or position encoder) is invalid **Error correction** - Check the configuration in the safe machine parameters and correct them if necessary - Switch the control off and on again - If the values of the machine parameters have been changed, they are to be loaded by entering the manufacturer's password. Then a corresponding acceptance test must be run. - Inform your service agency.

Error number	Description
330-00B3	**Error message** MC self test %1 not started **Cause of error** The self-test for safety detected an error. A certain signal must be set for test purposes. This signal was not released by the indicated HSCI component. The error message contains the following information: MC self test STEST_<signal>, STESTDEV_<HSCI component>, <HSCI address> - The signal to be set is indicated under STEST_. - The HSCI component that did not set the signal is indicated under STESTDEV_. - The given number in the error message matches the HSCI address of the HSCI component concerned. Possible causes: - HSCI component is defective. - Wiring is faulty. **Error correction** - Exchange the HSCI component. - Check the wiring - Inform your service agency
330-00B4	**Error message** MC self test %1 not detected **Cause of error** The self-test for safety detected an error. A certain signal was set and was not recognized by the indicated HSCI component. The error message contains the following information: MC self test STEST_<signal>, STESTDEV_<HSCI component>, <HSCI address> - The set signal is indicated under STEST_. - The HSCI component that did not recognize the set signal is indicated under STESTDEV_. - The given number in the error message matches the HSCI address of the HSCI component concerned. Possible causes: - HSCI component is defective. - Wiring is faulty. **Error correction** - Exchange the HSCI component. - Check the wiring - Inform your service agency.
330-00B5	**Error message** MC Incorrect command from the MC **Cause of error** - Internal software error **Error correction** - Inform your service agency

Error number	Description
330-00B6	**Error message** MC Invalid SPLC operating mode **Cause of error** - Error in SPLC program - SPLC requests an invalid operating mode The valid modes are SOM_1, SOM_2, SOM_3, SOM_4 - Internal software error **Error correction** - Check the SPLC program - Inform your service agency
330-00B7	**Error message** MC brake control is defective %2 **Cause of error** - Brake output of the inverter is defective - PL output for controlling the brake is defective - Incorrect wiring of the braking control (e.g. short circuit to 0 V, short circuit to 24 V) **Error correction** - Check the wiring of the brake control - Check the wiring for short circuits to 0 V or 24 V - Check the supply voltage of the outputs to the brake control (PL module, inverter) - Inform your service agency.
330-00B8	**Error message** MC brake control is defective **Cause of error** - Brake output of the inverter is defective - PL output for controlling the brake is defective - Incorrect wiring of the braking control (e.g. short circuit to 0 V, short circuit to 24 V) **Error correction** - Check the wiring of the brake control - Check the wiring for short circuits to 0 V or 24 V - Check the supply voltage of the outputs to the brake control (PL module, inverter) - Inform your service agency.

Error number	Description
330-00B9	**Error message** MC error in the cutout channel STO.A.x (-STO.A.P.x=0) %2 **Cause of error** The cutout channel test detected a fault: The axis-specific pulse deletion for the power stage over the A channel is defective. Switch-off signal: STO.A.P.x Switch-off signal: STO.A.x The acknowledgment signal for this cutout channel has the wrong condition, i.e. when - JH inverter: Power stage reports "readiness" (RDY.x=1), although "No readiness" (RDY.x=0) is expected. - DRIVE-CLiQ inverter: Associated diagnostic signal is "1", although "0" is expected. **Error correction** - Check the wiring (PWM cable) - Hardware error (power module, controller unit) - Inform your service agency.
330-00BA	**Error message** MC error in the cut-out channel STO.A.x (-STO.A.P.x=1) %2 **Cause of error** The cutout channel test detected a fault: The axis-specific pulse release for the power stage over the A channel is defective. Switch-off signal: STO.A.P.x Switch-off signal: STO.A.x The acknowledgment signal for this cutout channel has the wrong condition, i.e. when - JH inverter: Power stage reports "no readiness" (RDY.x=0), although "readiness" (RDY.x=1) is expected. - DRIVE-CLiQ inverter: Associated diagnostic signal is "0", although "1" is expected. **Error correction** - Check the wiring (PWM cable) - Hardware error (power module, controller unit) - Inform your service agency.

Error number	Description
330-00BB	**Error message** CC%2 error in cut-out channel STO.A.x (-STO.A.MC.WD=0) ax-grp A **Cause of error** The cutout channel test detected a fault: The CC-specific pulse deletion for the corresponding power modules of the NC axis group (A) over the A channel is defective. Switch-off signal: STO.A.MC.WD Switch-off signal: STO.A.x At least one of the acknowledgment signals for this cutout channel has the wrong condition, i.e. when - JH inverter: At least one power module reports "readiness" (RDY.x=1), although for all power modules of the axis group A of the CC, "No readiness" (RDY.x=0) is expected. - DRIVE-CLiQ inverter: At least on of the associated diagnostic signal is "1", although "0" is expected. **Error correction** - Check the wiring (PWM cable) - Hardware error (power module, controller unit) - Inform your service agency.
330-00BC	**Error message** CC%2 error in cut-out channel STO.A.x (-STO.A.MC.WD=0) ax-grp. S **Cause of error** The cutout channel test detected a fault: The CC-specific pulse deletion for the corresponding power modules of the spindle axis group (S) over the A channel is defective. Switch-off signal: STO.A.MC.WD Switch-off signal: STO.A.x At least one of the acknowledgment signals for this cutout channel has the wrong condition, i.e. when - JH inverter: At least one power module reports "readiness" (RDY.x=1), although for all power modules of the axis group S of the CC, "No readiness" (RDY.x=0) is expected. - DRIVE-CLiQ inverter: At least on of the associated diagnostic signal is "1", although "0" is expected. **Error correction** - Check the wiring (PWM cable) - Hardware error (power module, controller unit) - Inform your service agency.

Error number	Description
330-00BD	**Error message** CC%2 error in cut-out channel STO.A.x (-STO.B.P.x) axis group A **Cause of error** The cutout channel test detected a fault: The CC-specific pulse release for the corresponding power modules of the NC axis group (A) over the A channel is defective. The ready signal RDY for one axis group is missing. The cutout channel STO.A.x was switched to inactive for the test of the cutout channel STO.B.x for all power modules. At least one of the acknowledgment signals for this cutout channel has the wrong condition, i.e. when - At least one power module reports "No readiness" (RDY.x=0), although "Readiness" (RDY.x=1) is expected for all power modules of the axis group of the CC. **Error correction** - Check the wiring (e.g. pulse release X72 of the supply module, X73 Heidenhain interface PCB, PWM cable is defective) - Exchange defective hardware (power module) - Inform your service agency
330-00BE	**Error message** CC%2 error in cut-out channel STO.A.x, RDY signal missing **Cause of error** The cutout channel test detected a fault: The CC-specific pulse release for the corresponding power modules of the spindle (S) over the A channel is defective. The ready signal RDY for one axis group is missing. The cutout channel STO.A.x was switched to inactive for the test of the cutout channel STO.B.x for all power modules of the axis group S. At least one of the acknowledgment signals for this cutout channel has the wrong condition, i.e. when - At least one power module reports "No readiness" (RDY.x=0), although "Readiness" (RDY.x=1) is expected for all power modules of the axis group of the CC. **Error correction** - Check the wiring (e.g. pulse release X71 of the supply module, X73 Heidenhain interface PCB, PWM cable is defective) - Exchange defective hardware (power module) - Inform your service agency

Error number	Description
330-00C0	**Error message** Self-test required **Cause of error** - Maximum permissible time interval for the self-test has been exceeded - For further operation of the machine with open guard doors, the self-test must be conducted **Error correction** - Start the self-test - If the guard doors are closed, the error message can be acknowledged and machine operation continued - With open guard doors, or before the guard doors are opened, the self-test must be conducted in order to continue operating the machine - Inform your service agency
330-00C1	**Error message** MC drives cannot be switched on: NN_GenSafe = 0 **Cause of error** - SPLC interface signal NN_GenSafe = 0. It is therefore impossible to switch on the drives. - SPLC program does not set the interface signal. **Error correction** - Check the SPLC program. - Inform your service agency.
330-00C2	**Error message** Error during activation of an FS configuration **Cause of error** The activation of another FS configuration was canceled because the CRC checksums of the FS data records differ. **Error correction** - Reset the "accepted" status of the most recently changed FS data records - Manually undo your most recently made changes - Copy a valid backup onto the machine - Inform your service agency
330-00C3	**Error message** FS configuration error: machine IDs do not match **Cause of error** The activation of another FS configuration was canceled because the entered machine IDs of the FS configurations differ. **Error correction** – Check the machine IDs of the FS configurations - Inform your service agency

Error number	Description
330-00C4	**Error message** Maximum number of FS data records has been reached **Cause of error** The activation of another FS configuration was canceled because the maximum permissible number of different FS data records was exceeded. **Error correction** - Delete any unneeded FS data records - Inform your service agency
330-00C5	**Error message** Maximum number of FS configurations has been reached **Cause of error** The activation of another FS configuration was canceled because the maximum permissible number of different FS configurations was exceeded. **Error correction** - Delete any unneeded FS configurations - Inform your service agency
330-00C6	**Error message** MC manual operation. Only one axis allowed. **Cause of error** — Two or more axes are being moved in the "Electronic Handwheel" operating mode. - Simultaneous movement of multiple axes is not allowed. **Error correction** - Move only one axis in the Handwheel operating mode. - Check the entry in the appropriate safe machine parameter and correct it if necessary. - Inform your service agency.
330-00C7	**Error message** A fatal reconfiguration error was triggered **Cause of error** - The reconfiguration process for functional safety (FS) has failed. **Error correction** - The status "accepted" will be reset for all configurations and data records for functional safety. - The comparison data records will be deleted. - Install a complete backup of the machine. - Inform your service agency.

Error number	Description
330-00C8	**Error message** Acceptance not allowed during configuration process **Cause of error** - Acceptance testing of data records configurations was conducted during an FS configuration process. This is not allowed. **Error correction** - Conduct the acceptance testing when the configuration process is completed. - Inform your service agency.
330-00C9	**Error message** Error during activation of an FS configuration **Cause of error** One of the following safe machine parameters was changed after the self-test began: - Time to next self-test - Time monitoring for brake test These machine parameters can be changed only before starting the safety self-test. **Error correction** - Undo the change in the affected machine parameters. - Restart the control. - Inform your service agency.
330-00CA	**Error message** Axis-group-specific guard door is open (MC) %1 **Cause of error** - Guard door of the axis group is opened The guard door has to be closed in order to conduct the safety self-test or the brake test of a respective axis group Further possible causes: - Faulty wiring of guard doors - Guard door contact is defective - SPLC interface signal PP_AxGrpStateReq unequal to S_S-TATE_AUTO [10] **Error correction** - Close the axis-group-specific guard doors in order to conduct the safety self-test or the brake test Further possible measures: - Check the wiring of the guard door contacts - Check the guard door contact - Check the SPLC program - Inform your service agency

Error number	Description
330-00CB	**Error message** Axis-group-specific guard door is open (CC) %1 **Cause of error** - The guard door is open The guard door(s) has to be closed in order to conduct the safety self-test or the brake test of an axis Further possible causes: - Faulting wiring of the guard door - Guard door contact is defective **Error correction** - Close the guard door(s) in order to conduct the safety self-test or the brake test Further possible measures: - Check the wiring of the guard door contacts - Check the guard door contact - Check the SPLC program - Inform your service agency
330-00CC	**Error message** The time until the next brake test is invalid for one axis %1 **Cause of error** - A time greater than 0 is configured for a non-safe axis **Error correction** - For an axis that is not monitored by functional safety (FS) , only 0 is allowed as the time value.
330-00DE	**Error message** Commissioning function for FS is active **Cause of error** Commissioning support for functional safety (FS) functions are active: - The NC software does not limit velocity - Unexpected movements or dangerous situations might occur **Error correction** - The machine must be operated only by trained personnel. - Operate the machine only with great caution - This function must be activated only for commissioning purposes - Deactivate this function before shipping the machine

Error number	Description
330-00E7	**Error message** FS data record cannot be accepted **Cause of error** - There are at least two FS data records with identical ID in different parameter sets and at least one safe parameter SMP has a different value in both parameter set. **Error correction** - Compare and adjust the values of the safe parameters SMP of the same data record between the parameter sets. - To avoid such errors, HEIDENHAIN recommends using the "KeySynonym" function. - Inform your service agency.
330-00E8	**Error message** Self-test required **Cause of error** - Maximum permissible interval time for the self-test has been exceeded - The self-test must be conducted for further operation of the machine with open guard doors **Error correction** - Start the self-test - When the guard doors are open, or before opening the guard doors, the self-test must be started in order to continue operating the machine
330-00EA	**Error message** Activation of the automatic change mode not possible **Cause of error** An FS reconfiguration has not yet concluded. **Error correction** Wait until the FS reconfiguration process is completed and then retry.

Error number	Description
330-00EC	**Error message** MC: S status reaction is active: %1 **Cause of error** An error bit was set in the S status by an internal hardware or software error: -SCC.B.WD: Watchdog WD.B.CC of a CC controller unit has timed out -SMOP.WD: Watchdog WD.A.SMOP or WD.B.SMOP of an MB or TE machine operating panel has timed out -SPL.WD: Watchdog WD.A.SPL or WD.B.SPL of a PLB has timed out -PF.BOARD: The internal voltage monitoring of an HSCI component has detected a faulty supply voltage -REQ.SS2: The internal temperature or fan monitoring of an HSCI component has detected a fault **Error correction** - Generate the service file - Inform your service agency
330-00ED	**Error message** MC error, device-specific evaluation CC %2 / %3, error %1 **Cause of error** The CC controller unit reports a device-specific error. **Error correction** Inform your service agency
330-00EF	**Error message** SMC error in the configuration data %1 **Cause of error** The configuration parameters do not match the expected values. **Error correction** Check the machine parameters and correct as needed
330-00F0	**Error message** Checking of axes of various axis groups **Cause of error** You attempted to check axes of different axis groups simultaneously. **Error correction** - Check the configuration: axes that are coupled in a rigid gantry combination must belong to the same axis group - Uncouple a dynamic gantry combination if you want to check it - If necessary, inform your machine manufacturer

Error number	Description
330-00F1	**Error message** Reconfiguration while checking the axes **Cause of error** Safe machine parameters were reconfigured while the axes were being checked. Checking was therefore aborted. **Error correction** - Re-check the axes - If the problem persists, inform your machine manufacturer
330-00F2	**Error message** Checking of an externally monitored axis **Cause of error** You attempted to check an externally monitored axis. The control can check only internally monitored axes. **Error correction** - Check the parameter CfgAxParSafety/encoderForSafety - Inform your service agency
330-00F3	**Error message** Permissive button missing while checking **Cause of error** While checking an axis, you failed to press the permissive button within the time prescribed by the control. **Error correction** Re-check the axis.
330-00F4	**Error message** Checking during a fatal error **Cause of error** You attempted to check an axis while the functional safety was in a fatal error state. **Error correction** - Restart the control - If necessary, inform your service agency

Error number	Description
330-00F5	**Error message** Axis not at test position **Cause of error** - The axis to be checked is not located at the test position (safe machine parameter positionMatch in CfgAxisSafety) - Axis is too far from the test position (safe machine parameter positionDiffRef in CfgAxisSafety) **Error correction** - Press CE to acknowledge the error message and move the axis to the test position - Then check the axis If the message appears although the axis is at the correct test position: - For gantry combinations, an axis other than the one being checked might not be in the correct position. If necessary, uncouple the gantry combination for checking. - Check the configuration of the axis traverse direction and correct it if necessary (machine parameter signCorrActualVal, signCorrNominalVal, or entry in the DIR column of the motor table) - Inform your machine manufacturer
330-00F6	**Error message** FS configuration with a parameter whose value cannot be accepted **Cause of error** Acceptance of CfgSafety/CfgAxParSafety > speedPosComp-type with the value noComp is not allowed. **Error correction** - Reset the acceptance status - Reset the parameter value
330-00F7	**Error message** Axis in motion **Cause of error** Possible causes: - The axis to be checked is still in motion - Another axis that belongs to the same axis group as the axis to be checked is still in motion **Error correction** - Press CE to acknowledge the error message and bring the axis to a standstill - Then check the axis - Inform your service agency

Error number	Description
330-00F8	**Error message** Internal software error **Cause of error** You checked an axis while another axis was still being checked **Error correction** - Conclude checking of the first axis before checking another axis - Inform your service agency
330-00F9	**Error message** Checking of axes of different axis groups **Cause of error** You checked two or more axes at the same time even though they belong to different axis groups **Error correction** - Make sure that all internally monitored axes of a gantry combination belong to the same axis group - Inform your machine manufacturer - Inform your service agency
400-0720	**Error message** Selected block not addressed Current block (%1) not selected **Cause of error** After an interruption in the program run, the control cannot resume program run from the present cursor location. **Error correction** Select the desired location for resuming the program with "GOTO" + block number, or with the mid-program startup function.
400-073E	**Error message** Parameter not found %1 **Cause of error** A value could not be read from the configuration data. **Error correction** Check the configuration data.
400-075F	**Error message** Error while reading the model data from %1 **Cause of error** Error while reading the model data. **Error correction** Delete the faulty file and make a new one.

Error number	Description
400-0760	**Error message** Error while writing the model data to %1 **Cause of error** Error while writing the model data. **Error correction** Check the memory capacity. Error in file system.
400-0761	**Error message** Directory '%1' could not be created. **Cause of error** Directory could not be created. **Error correction** Check the memory capacity. Error in file system.
400-0768	**Error message** Selection of %1 not allowed **Cause of error** The tool table tool.t is meant only for program run. **Error correction** Select another tool table.
400-0773	**Error message** Unable to write parameter **Cause of error** A configuration file might be write protected. **Error correction**
400-0775	**Error message** Incomplete graphic model **Cause of error** **Error correction**
400-077F	**Error message** File is being saved and cannot yet be opened **Cause of error** The file is now being saved in the editor. **Error correction** - After the saving process is complete, reselect the file. - The condition is indicated by the “Please wait” icon.

Error number	Description
401-0001	**Error message** Message %1 cannot be transmitted **Cause of error** Internal software error **Error correction** Inform your service agency.
401-0002	**Error message** The application could not be initialized **Cause of error** Logon with the confiration server is not possible. Configuration of the programmable axes is contradictory. **Error correction** Check the configuration data and edit.
401-0003	**Error message** Bad option %1 specified **Cause of error** Internal software error **Error correction** Inform your service agency.
401-0004	**Error message** The file %1 already contains a binary coded NC program **Cause of error** **Error correction**
401-0005	**Error message** The file %1 does not contain an NC program **Cause of error** **Error correction**
401-0006	**Error message** The file %1 already contains an encrypted NC program **Cause of error** **Error correction**
401-0007	**Error message** The file %1 contains data in an unknown format **Cause of error** **Error correction**

Error number	Description
401-0008	**Error message** The file %2 will be overwritten by file %1 **Cause of error** **Error correction**
401-0009	**Error message** Specified drive %2 of the file %1 cannot be substituted **Cause of error** An NC program is to be installed on a drive other that 0:, R: or V:. **Error correction** Check and edit the configuration data for cycles and NC macros.
401-000A	**Error message** %2 file was converted (%1) **Cause of error** **Error correction**
401-000B	**Error message** File %1 could not be converted to file %2 **Cause of error** Error during conversion of an NC program. The file could not be saved. **Error correction** Check the path name and write protection of the target file.
401-000C	**Error message** File %2 was encrypted **Cause of error** **Error correction**
401-000D	**Error message** Installation of cycles complete **Cause of error** **Error correction**
401-000E	**Error message** Error: **Cause of error** **Error correction**

Error number	Description
401-000F	**Error message** %2 file was converted **Cause of error** **Error correction**
401-0010	**Error message** %1 file contains data in an illegible format **Cause of error** At an earlier time the program was converted to an unreadable format. No backup copy was made from which another conversion would be possible. **Error correction** Recopy the file with the source text of the program to the control and start the conversion again.
401-0011	**Error message** The file %1 contains syntactically incorrect NC block %2. **Cause of error** The program uses an unknown cycle or axis, or contains other syntax errors. **Error correction** Copy the file with the corrected program source text to the control again and restart the conversion.
402-0001	**Error message** FK programming: Contradictory input **Cause of error** You programmed contradictory data within a contour element or in different contour elements. **Error correction** Check the entered data and modify it.
402-0002	**Error message** FK programming: Undefined starting position **Cause of error** You did not define an unambiguous tool position before beginning an FK sequence. **Error correction** Before beginning an FK sequence, program a positioning block with both coordinates of the working plane.

Error number	Description
402-0003	**Error message** FK programming: No FPOL defined **Cause of error** You programmed polar coordinates within an FK sequence without first defining the pole. **Error correction** Use the FPOL function to program a pole.
402-0004	**Error message** FK programming: FSELECT not allowed. **Cause of error** An FK sequence contains an FSELECT block although the contour is already clearly defined. **Error correction** Edit the NC program: delete the corresponding FSELECT block.
402-0005	**Error message** FK programming: Contour too complex **Cause of error** The number of unresolvable FK blocks or the number of selectable alternative contours exceeds the permissible maximum value of 32 each. **Error correction** Use FSELECT to resolve the FK sequence earlier, or enter additional data.
402-0006	**Error message** FK programming: Internal software error **Cause of error** The control software could not calculate the programmed contour, although it satisfies all formal requirements tested. **Error correction** Try to program the desired contour in another way. If necessary, inform the service agency for you control.
402-0007	**Error message** FK programming: Illegal coordinate **Cause of error** You programmed an illegal axis within an FK sequence. **Error correction** Program only coordinates in the working plane that you defined using FPOL (default: XY plane).

Error number	Description
402-0008	**Error message** FK programming: Incomplete input **Cause of error** You did not program all required data within an FK sequence. The following are illegal: only one coordinate in the FPOL block, only one coordinate of a auxiliary point (PD, P1, P2 or P3), PD auxiliary point without the distance DP or vice versa, FC/FCT circular arc without definition of a rotational direction (DR), the distance of a contour to a parallel line (DP) without the parallel line (PAR) or vice versa. **Error correction** Add the missing data to the NC program.
402-0009	**Error message** FK programming: Illegal positioning block **Cause of error** Within an unresolved FK sequence you programmed an illegal positioning block other than FK blocks, RND/CHF, APPR/DEP, and L blocks with motion components exclusively perpendicular to the FK plane. **Error correction** First resolve the FK sequence completely or delete illegal positioning blocks. Geometry functions that are defined over the gray contouring keys and have coordinates in the working plane are illegal (exception: RND, CHF, APPR/DEP).
402-000A	**Error message** FK programming: Illegal operation **Cause of error** The following are illegal in open FK sequences: PGM END (program end), CYCL DEF 7-11 and 26 (coordinate transformations), TOOL CALL (tool change), and PGM CALL (program calls) **Error correction** First resolve the FK sequence or program the objectionable operation at another location in the program.

Error number	Description
402-000B	**Error message** FK programming: Illegal block reference **Cause of error** From an FK sequence, a block number is used to make reference to the end point or the end tangent of a block that either - does not exist - is too far away (> 64 blocks below or > 32 blocks above) - is not a positioning block - belongs to a category of positioning blocks which cannot be used for references (transitions, CC, FPOL, blocks containing only axis values or machine coordinates) **Error correction** Enter a reference to another block, or do without the reference
402-000C	**Error message** FK programming: Incomplete input **Cause of error** The FK sequence is not complete at its program end. You will have to program additional data or NC blocks. **Error correction** * Add FK blocks as the end or edit them. * Add the missing data or NC blocks within the program. Note: - In each block, program the data that do not change. (Non-programmed data are treated as unknown.) - If the first block of an FK contour is an FCT or FLT block, you must program at least two NC blocks with the gray path function keys to fully define the direction of contour approach. An FK contour must not be programmed immediately after an LBL command
600-0009	**Error message** Casting machining was interrupted due to an internal error. Machining is not possible. %1 **Cause of error** Data overflow or insufficient memory. **Error correction** Check the program. If necessary, correct the finished part.

Error number	Description
600-000A	**Error message** Blank defined smaller than finished part. %1 **Cause of error** Error in the part definition. **Error correction** Redefine the table.
600-000B	**Error message** Drill tip longer than bore! No rough drilling takes place. %1 **Cause of error** Perhaps the inside contour resulted unintentionally from contour generation of the finished part. **Error correction** Check the program. If necessary, correct the finished part.
600-000C	**Error message** No free memory capacity. %1 **Cause of error** The data module is too small or the contours are too large. **Error correction** If possible, simplify the part.
600-000F	**Error message** Internal error - more information in the system warning %1 **Cause of error** **Error correction**
600-0011	**Error message** There is nothing to machine or nothing can be machined under these preconditions. %1 **Cause of error** Error in the data. **Error correction** Redefine the program.

Error number	Description
600-0012	**Error message** No automatic function available for this function. Do the machining manually. %1 **Cause of error** Incorrect main machining mode. **Error correction** Redefine the main machining mode.
600-0013	**Error message** Since "Clamping" was not used, an external chuck with 15 mm clamping length is assumed. %1 **Cause of error** The part is not clamped. **Error correction** Interrupt TURN PLUS and clamp the part.
600-0015	**Error message** Since no cutting limit was defined, it will be set to the end of the workpiece %1 **Cause of error** The cutting limit assignment will be exited without data. **Error correction** Clamp or enter the values for the cutting limit.
600-0031	**Error message** Tool %2 was programmed without cutting speed. A default value is set. %1 **Cause of error** The database is incomplete. **Error correction** Expand the cutting database.
600-0032	**Error message** Tool %2 was programmed without main feed! A default value is set. %1 **Cause of error** The database is incomplete. **Error correction** The cutting database must be expanded.

Error number	Description
600-0033	**Error message** You are attempting to load cutting data for an unidentifiable tool number. This is not possible. %1 **Cause of error** Since the tool was already loaded via DCS, the cause can only be a software or hardware error. **Error correction** Load the part again and restart AWG.
600-0034	**Error message** No cutting data available for this tool. (Tool number, Material, Cutting material %2) %1 **Cause of error** The database has no data beyond the current combination of cutting material. **Error correction** Expand the cutting database, then restart the AWG.
600-0041	**Error message** Defined secondary machining mode is invalid! %1 **Cause of error** Software error or incorrect data in the machining sequence. **Error correction** Check the entered machining sequence.
600-0049	**Error message** Generated working block contains incorrect record types or technology record is missing. %1 **Cause of error** No technology record was made. **Error correction** Check whether a tool was selected.
600-0051	**Error message** Due to a tool geometry cutting limitation an area to be finished cannot be machined. %1 **Cause of error** The tool diameter is too large. **Error correction** Select or define another tool.

Error number	Description
600-0081	**Error message** For automatic calculation of the tool change position, tool selection must be set to turret! %1 **Cause of error** The parameter for tool selection is set incorrectly. **Error correction** In the parameter editor, set the tool selection to turret.
600-0083	**Error message** No valid slide number was found, machining is taking place with slide number 1! %1 **Cause of error** Header is invalid. **Error correction** The slide number must be entered in the header.
600-00A9	**Error message** No data on machining site; no automatic tool selection possible! %1 **Cause of error** The data were not entered correctly. **Error correction** Reenter the data.
600-00AA	**Error message** No data on machining direction; no automatic tool selection possible! %1 **Cause of error** The data were not entered correctly. **Error correction** Reenter the data.
600-00AB	**Error message** No data on machining mode; no automatic tool selection possible! %1 **Cause of error** The data were not entered correctly. **Error correction** Reenter the data.

Error number	Description
600-00B1	**Error message** Since no drill with sufficient effective length is available, no through-drilling is possible. %1 **Cause of error** No suitable tool available. **Error correction** If possible, add to the data bank.
600-00B2	**Error message** Since no suitable internal roughing tool was found, only a partial section can be machined. %1 **Cause of error** No suitable tool available. **Error correction** If possible, add to the data bank.
600-00B3	**Error message** Since no internal finishing tool was found, only a partial section can be machined. %1 **Cause of error** No suitable tool available. **Error correction** If possible, add to the data bank.
600-00B4	**Error message** No suitable tools available (Ideal-/Alternative-/Emergency tool: %2)! %3 %1 **Cause of error** The database is too small. **Error correction** Enter more tools, or increase the possible tolerance values.
600-00B5	**Error message** Unknown tool type defined! No automatic tool selection possible. %1 **Cause of error** Tool type input error. **Error correction** Reenter the tool type.

Error number	Description
600-00CC	**Error message** The 1st hole limit diameter must not be smaller than the 2nd hole limit diameter. %1 **Cause of error** UBD2 > UBD1 **Error correction** Exchange the diameter values.
600-00F9	**Error message** Due to technological cutting limitations, specific machining areas must be omitted. %1 **Cause of error** Cutting limitations of the chuck. **Error correction** Clamp the part differently.
600-0101	**Error message** No tool selection possible with available data on machining mode, site and direction! %1 **Cause of error** The data were not entered correctly. **Error correction** Reenter the data.
600-0149	**Error message** Elements with unknown machining data were found in the finished part contour. %1 **Cause of error** Error in the analysis. **Error correction** Reload the contour and restart the AWG.
600-0181	**Error message** Starting or overtravel length of thread too long! Collision with workpiece or chuck jaws! %1 **Cause of error** Starting length or overtravel length is defined too large. **Error correction** Correct the starting or overtravel length.

Error number	Description
600-0189	**Error message** Secondary machining direction of tool invalid; end face elements are not finished faced. %1 **Cause of error** Auxiliary cutting edge undefined or incorrectly defined. **Error correction** Define the auxiliary cutting edge correctly.
600-01C3	**Error message** No attributes defined for thread. %1 **Cause of error** No thread attributed defined. **Error correction** Define thread attributes.
600-01C4	**Error message** Contour contains thread whose edge elements are not finish machined. %1 **Cause of error** Unmachined contour areas. **Error correction** Machine manually or after a second clamping.
600-01C9	**Error message** More than 6 threads were defined within a machining site. %1 **Cause of error** More that six threads were defined for one machining location. **Error correction** Define a maximum of six threads per machining location.
600-0211	**Error message** Finished part does not line entirely within the workpiece blank. Defined part can't be machined in this condition. %1 **Cause of error** The parts were entered incorrectly. **Error correction** Delete one part and define it again.

Error number	Description
600-0212	**Error message** Contour sense of rotation undeterminable! %1 **Cause of error** The parts were entered incorrectly. **Error correction** Delete the part and define them again.
600-0213	**Error message** Blank contour contains irreparable errors! Machining is not possible. %1 **Cause of error** The part was entered incorrectly. **Error correction** Delete the part and define it again.
600-0214	**Error message** Finished part contour contains irreparable errors! Machining is not possible. %1 **Cause of error** The part was entered incorrectly. **Error correction** Delete the part and define it again.
600-0215	**Error message** No program header defined! Definition required prior to machining. %1 **Cause of error** The program header was not generated or it has an old structure. **Error correction** Generate a program head.
600-0229	**Error message** Corner is unsuitable for the separation point. Cond.: inner corner angle > 180 deg. - inward copy angle %1 **Cause of error** The AWG cannot follow the machining strategy resulting from the chosen separation point. **Error correction** Either machine the part with the interactive working plane generation (IWG) or change the separation point.

Error number	Description
600-022A	**Error message** Contour element for machining through the separation is within clamping range. Separation point is deleted. %1 **Cause of error** The separation point lies within the clamping range. **Error correction** Either clamp differently or change the separation point.
600-0239	**Error message** Either all secondary machining modes must be defined or none. %1 **Cause of error** **Error correction**
600-0261	**Error message** Pocket/island milling is not yet possible. Alternatively the contour is milled. %1 **Cause of error** You defined a contour without attributes. You did not define whether it is machined internally or externally, therefore pocket milling is assumed. **Error correction** Assign contour attributes.
600-0262	**Error message** The cutter radius cannot be determined from the contour. A default cutter is used (%2). %1 **Cause of error** The contour now has circular inside corners. The cutter diameter cannot be determined. **Error correction**
600-0263	**Error message** Contour sense of rotation undetermined. The cutter radius cannot be determined ==> standard cutter (%2 dia.) %1 **Cause of error** The contour is open. Open contours can only be engraved. **Error correction**

Error number	Description
600-0264	**Error message** Machining side is not clear. The contour side is selected depending on the depth. %1 **Cause of error** There were no contour attributes assigned. **Error correction** Assign contour attributes.
600-0265	**Error message** Pockets can only be machined internally. %1 **Cause of error** You assigned either the "contour" or "external" attribute to the pocket. **Error correction** Assign the "internal" attribute.
600-0266	**Error message** Islands can only be machined externally. %1 **Cause of error** You assigned either the "contour" or "internal" attribute to the island. **Error correction** Assign the "external" attribute.
600-0267	**Error message** Tool with the defined %2 diameter was not found. Select tool with a smaller diameter %3. %1 **Cause of error** The corresponding tool is not in the turret or in the file, or the tool with the given diameter is not well suited for the task. **Error correction** Make the tools available or change the parameter.
600-02D9	**Error message** No tool found. Use a left-hand tool to machine the front side and a right-hand tool to machine the rear. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.

Error number	Description
600-02DA	**Error message** No tool found in the file with which machining is to be carried out. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters.
600-02DB	**Error message** No tool found in the file which can be used to carry out the machining process. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02DC	**Error message** No tool found with a permissible secondary machining direction needed for machining. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02DD	**Error message** No tool found. Contour cannot be machined using the adjustment and tool tip angle specified %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.

Error number	Description
600-02DE	**Error message** No tool found. The tool cutting width (without radius) is too large for the recess. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02DF	**Error message** No tool found. The plunging depth of the tool is insufficient. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02E0	**Error message** No tool found whose direction of rotation fits the machining side of the contour and the milling direction. %1 **Cause of error** A tool is being sought whose a direction of rotation that has not yet been defined. **Error correction** In the machining attributes, change up-cut to climb, or vice versa.
600-02E1	**Error message** No tool found. Machining is only possible with stationary tools. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.

Error number	Description
600-02E2	**Error message** No tool found. Machining is only possible with driven tools. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02E3	**Error message** No tool found. Please check the milling machine cutting teeth. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02E4	**Error message** No tool found. The effective length is insufficient %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02E5	**Error message** No tool found. Please check the milling diameter. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.

Error number	Description
600-02E6	**Error message** No tool found. Please check the milling angle. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02E7	**Error message** Multi-cut tools are not supported and are therefore rejected. %1 **Cause of error** The automatic tool selection does not work for multi-cut tools. **Error correction**
600-02E8	**Error message** No tool found that also fits in the tool mount location. %1 **Cause of error** The mount types of tool and tool carrier do not match. **Error correction** If the mount type is incorrect, adjust the mount type of the tool to the mount type of the tool-carrier mount positions.
600-02E9	**Error message** No tool found. The effective length is insufficient %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02EA	**Error message** No tool found. Please check drill diameter. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.

Error number	Description
600-02EB	**Error message** No tool found. Please check tool tip angle. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02EC	**Error message** No tool found. Please check tappet diameter. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02ED	**Error message** No tool found. Please check tappet length. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02EE	**Error message** No tool found. Check countersinking angle. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.

Error number	Description
600-02EF	**Error message** No tool found. Please check thread pitch. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02F0	**Error message** No tool found. Please check centering angle. %1 **Cause of error** Optimization deletes all tools because they seem unsuitable for the AWG. **Error correction** Check the parameters stored under tool geometry with those of the tool.
600-02F1	**Error message** No tool found. Data on cutting materials are missing. %1 **Cause of error** There are no cutting materials entered on the tool data. **Error correction** Enter the cutting materials in the tool data.
600-0301	**Error message** Selection parameters for tool selection mode not set! %1 **Cause of error** Parameter error. **Error correction** In the parameter editor, set the parameter for type of tool selection.
600-0302	**Error message** No tool and turret data defined for entered slide number. %1 **Cause of error** Parameter error. **Error correction** In the parameter editor, set the parameter for type of tool selection.

Error number	Description
600-0304	**Error message** Machining only possible using the turret! %1 **Cause of error** The automatic magazine position assignment is not supported by the AWG. **Error correction** In the parameter editor, set the parameter for type of tool selection.
600-0305	**Error message** Invalid slide/spindle assignment in program header! %1 **Cause of error** Incorrect parameter description in the program header. **Error correction** Change the program header.
600-0306	**Error message** Invalid slide number specification in program header! %1 **Cause of error** Incorrect parameter description in the program header. **Error correction** Change the program header.
600-0307	**Error message** Mount type of tool %2 does not fit any mount type in turret. %1 **Cause of error** During automatic tool selection from a file, the control compares the tool mount types listed for the turret with the tool mount types entered in the NC program. The types must match. **Error correction** Ensure that the tool holder types match.
600-0309	**Error message** Selected tool not in turret. %1 **Cause of error** The turret was incorrectly assigned, or the tool selection parameter is incorrectly set. **Error correction** Set the tool selection to "NEW" turret and start INI_REVBELE-GUNG.

Error number	Description
600-030A	**Error message** Turret is full! %1 **Cause of error** The tool turret is full. **Error correction** Check the turret assignment and remove superfluous tools.
600-030B	**Error message** Selection parameter for tool selection mode not set! %1 **Cause of error** Parameter error. **Error correction** In the parameter editor, set the parameter for type of tool selection.
600-030C	**Error message** Tool %2 is not suitable for mount location %3. %1 **Cause of error** Due to its mount type and preferred mount in the tool carrier description, the tool does not fit into its ideal, alternative, or emergency location. It is therefore simply placed in a vacant location. **Error correction** Edit the tool mount locations in the tool carrier description.
600-030D	**Error message** There are no longer any free tool mount locations or tool does not fit in carrier due to the mount type. %1 **Cause of error** The tool carrier is full, or the tool or the vacant locations in the carrier do not share the same mount types. **Error correction** If the mount type is incorrect, adjust the mount type of the tool to the mount type of the tool-carrier mount positions.

Error number	Description
600-030E	**Error message** No tool was found that also fits in the tool mount location. %1 **Cause of error** The mount types of tool and tool carrier do not match. **Error correction** If the mount type is incorrect, adjust the mount type of the tool to the mount type of the tool-carrier mount positions.
600-030F	**Error message** Tool %2 has no cam or location number. %1 **Cause of error** A cam number or code number is missing in the database. **Error correction** Enter a code or cam number in the tool data.
600-0379	**Error message** Tool not contained in data base. (Tool number %2)! %1 **Cause of error** The tool was inserted directly into the turret. **Error correction** Save the tool data in the database.
600-0399	**Error message** Shaft is not prepared; no reverse machining possible in this clamping mode. %1 **Cause of error** The shaft has a constant diameter. **Error correction** Delete the part and define it again.
603-0161	**Error message** Extension of limiting contour at traversing angle not possible due to formation of a contour loop. %1 **Cause of error** You selected an traversing angle at which the limiting contour intersects itself when extended in the traversing angle. **Error correction** Select a traversing angle that rules out a contour loop, or select a different machining range.

Error number	Description
603-0162	**Error message** Extension of limiting contour at travel angle not possible due to formation of a contour loop. %1 **Cause of error** You selected an departure angle at which the limiting contour intersects itself when extended in the travel angle. **Error correction** Select a travel angle that rules out a contour loop, or select a different machining range.
603-0164	**Error message** Error in finished part or blank definition. Starting element of BEA area outside blank. %1 Contour area cannot be executed with active tool. **Cause of error** There is probably a blank form definition error. This means that the limiting contour lies in the vicinity of the starting element outside the workpiece blank. **Error correction** Check the workpiece blank definition in the part program and correct it if necessary.
603-0165	**Error message** Error in finished part or blank definition. An end element of BEA area outside blank. %1 **Cause of error** This is probably a blank form definition error, which means that the limiting contour lies in the vicinity of the end element outside the workpiece blank. **Error correction** Check the workpiece blank definition in the part program and correct it if necessary.
603-0182	**Error message** Residual material due to cutting edge geometry! %1 **Cause of error** You selected a poorly suited tool. **Error correction** Select a tool whose geometry enables it to machine grooves in the contour.

Error number	Description
603-01A2	**Error message** The complete machining area is a recessing contour and was erased. %1 **Cause of error** The cycle was not intended to machine recessing contours. Because it identified the entire limiting contour as recessing contour, it was deleted. **Error correction** Either select the cycle "with recessing," or select another machining area.
603-01A3	**Error message** Too many recessing areas! Cycle not executable due to a lack of memory. Reduce machining area. %1 **Cause of error** The recessing areas are saved internally in a contour list. Since there was not enough memory to save all recessing areas, the cycle had to interrupt. **Error correction** Try to run the cycle with a smaller machining area.
603-01E0	**Error message** Invalid position of workpiece blank corner (X1,Z1) %1 **Cause of error** The workpiece blank corner was defined such that the workpiece does not fully include the ICP contour. **Error correction** Correct the coordinates (X1,Z1)
603-01E2	**Error message** Residual material in recess due to cutting edge width of recessing tool. %1 **Cause of error** Due to its cutting-edge geometry, the tool cannot reach every point in the selected machining area. **Error correction** Select another tool, or change the machining area.

Error number	Description
603-0242	**Error message** Nothing to machine for cycle in selected contour area. %1 **Cause of error** It could be that the oversize is greater than the maximum distance between limiting contour and workpiece blank contour. **Error correction** Select a smaller oversize or another machining area.
603-0243	**Error message** Nothing to machine for cycle in selected contour area. %1 **Cause of error** You selected oversizes that are greater than the max. distance between limiting contour and workpiece blank contour, or you programmed machining in an area that has already been machined. **Error correction** Select a smaller oversize or another machining range.
603-0244	**Error message** Longitudinal cutting limitation incompatible with current tool position. Position the tool. %1 **Cause of error** You defined a longitudinal cutting limitation and the tool is positioned so that the cutting limitation shields the limiting contour from the tool. **Error correction** The tool must be positioned on the same side of the cutting limitation as the limiting contour to be machined. Reposition the tool accordingly.
603-0245	**Error message** Transverse cutting limitation incompatible with current tool position. Position the tool. %1 **Cause of error** You defined a transverse cutting limitation and the tool is positioned so that the cutting limitation shields the limiting contour from the tool. **Error correction** The tool must be positioned on the same side of the cutting limitation as the limiting contour to be machined. Reposition the tool accordingly.

Error number	Description
603-0247	**Error message** Equidistant allowance contour cannot be calculated. Cycle cannot be executed. %1 **Cause of error** The equidistant replied with an error message and the equidistant was not executed. **Error correction** Inform your service agency.
603-0261	**Error message** Machining with the selected tool is impossible! Select another tool! %1 **Cause of error** The auxiliary machining direction, which results from the cutting edge geometry, is not entered in WZ_NORM. The use of the tool is therefore not permissible. **Error correction** Select another tool.
603-0281	**Error message** Tool position unknown! Position tool. %1 **Cause of error** You defined a longitudinal cutting limit. There are no valid position coordinates defined for the tool. **Error correction** Position the tool.
603-0282	**Error message** Tool position unknown! Position tool. %1 **Cause of error** A cutting limit plan was defined. There are no valid position coordinates defined for the tool. **Error correction** Position the tool.

Error number	Description
603-0283	**Error message** Tool positioned exactly on longitudinal cutting limitation. Machining area cannot be determined. %1 **Cause of error** The control cannot recognize which side of the limit along the limiting contour to machine, because the tool is located exactly on the limiting axis. **Error correction** Position the tool longitudinally on the side of the cutting limitation to be machined.
603-0284	**Error message** Tool positioned exactly on transverse cutting limitation. Machining area cannot be determined. %1 **Cause of error** The control cannot recognize which side of the limit along the limiting contour to machine, because the tool is located exactly on the limiting axis. **Error correction** Position the tool transversely on the side of the cutting limitation to be machined.
603-02A1	**Error message** Rough-machining cycle called with invalid tool! %1 **Cause of error** **Error correction** Select another tool.
603-02A2	**Error message** Recessing cycle called with invalid tool! %1 **Cause of error** **Error correction** Select another tool.
603-02A3	**Error message** Finish-machining cycle called with invalid tool! %1 **Cause of error** **Error correction** Select another tool.

Error number	Description
603-02A4	**Error message** Contour area cannot be machined with current tool either with main (+Z) or secondary cutting edge (+X). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02A5	**Error message** Contour area cannot be machined with current tool either with main (+Z) or secondary cutting edge (-Z). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02A6	**Error message** Contour cannot be machined with current tool either with main (+Z) or secondary cutting edge (-X). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02A7	**Error message** Contour area cannot be machined with current tool either with main (+X) or secondary cutting edge (+Z). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02A8	**Error message** Contour area cannot be machined with current tool either with main (+X) or secondary cutting edge (-Z). %1 **Cause of error** **Error correction** Select another contour area or another tool.

Error number	Description
603-02A9	**Error message** Contour area cannot be machined with current tool either with main (+X) or secondary cutting edge (-X). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02AA	**Error message** Contour area cannot be machined with current tool either with main (-Z) or secondary cutting edge (+Z). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02AB	**Error message** Contour area cannot be machined with current tool either with main (-Z) or secondary cutting edge (X). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02AC	**Error message** Contour area cannot be machined with current tool either with main (-Z) or secondary cutting edge (-X). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02AD	**Error message** Contour area cannot be machined with current tool either with main (-X) or secondary cutting edge (+Z). %1 **Cause of error** **Error correction** Select another contour area or another tool.

Error number	Description
603-02AE	**Error message** Contour area cannot be machined with current tool either with main (-X) or secondary cutting edge (+X). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02AF	**Error message** Contour area cannot be machined with current tool either with main (-X) or secondary cutting edge (-Z). %1 **Cause of error** **Error correction** Select another contour area or another tool.
603-02B0	**Error message** Machining direction incompatible with contour direction. Longitudinal or transverse cycle cannot be executed. %1 **Cause of error** A longitudinal or transverse cycle was called with a tool whose machining direction does not match the limiting contour direction. **Error correction** Select another tool or change the machining direction.
603-02B1	**Error message** Recess cannot be machined with selected tool. Check RECESS and TOOL SELECTION. %1 **Cause of error** Either you attempted to machine a recess with the width 0, or you attempted to machine a radial recess with an axial tool, or vice versa. **Error correction** Check the recess and the selected tool.
603-02C1	**Error message** Approach to limiting contour at this traversing angle invalid due to cutting edge geometry! %1 **Cause of error** With the programmed tool, the approach to the limiting contour is not permissible at this angle because otherwise the side rake would gouge the workpiece. **Error correction** Select another traversing angle or another tool.

Error number	Description
603-02C2	**Error message** Travel from limiting contour at this travel angle invalid due to cutting edge geometry. %1 **Cause of error** With the programmed tool, the departure from the limiting contour is not permissible at this angle because otherwise the side rake would gouge the workpiece. **Error correction** Select another travel angle or another tool.
603-02C3	**Error message** Transverse cutting limitation incompatible with current tool position. Position the tool. %1 **Cause of error** The present tool position and the selected longitudinal cutting limitation make it impossible to machine the limiting contour in the roughing cycle. **Error correction** Position the tool so that it is on the other side of the cutting limitation axis, or select another transverse cutting limitation.
603-02C4	**Error message** Longitudinal cutting limitation incompatible with current tool position. Position the tool. %1 **Cause of error** The present tool position and the selected longitudinal cutting limitation make it impossible to machine the limiting contour in the roughing cycle. **Error correction** Position the tool so that it is on the other side of the cutting limitation axis, or select another longitudinal cutting limitation.
603-02E1	**Error message** Longitudinal cutting limitation incompatible with current tool position. Position the tool. %1 **Cause of error** The present tool position and the selected longitudinal cutting limitation make it impossible to machine the limiting contour in the recessing cycle. **Error correction** Position the tool so that it is on the other side of the cutting limitation axis, or select another longitudinal cutting limitation.

Error number	Description
603-02E2	**Error message** Transverse cutting limitation incompatible with current tool position. Position the tool. %1 **Cause of error** The present tool position and the selected longitudinal cutting limitation make it impossible to machine the limiting contour in the recessing cycle. **Error correction** Position the tool so that it is on the other side of the cutting limitation axis, or select another transverse cutting limitation.
603-0301	**Error message** Insufficient memory for individual contour areas. Cycle cannot be executed. Reduce machining area. %1 **Cause of error** The limiting contour switches so often with the workpiece contour that the individual limiting contour areas cannot be saved in the contour list. **Error correction** Reduce the machining area and try again.
603-0321	**Error message** Incorrect tool for machining. Select WO 2 or WO 8. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select on of the proposed tools.
603-0322	**Error message** Incorrect tool for machining. Select WO 4 or WO 6. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.

Error number	Description
603-0323	**Error message** Incorrect tool for machining. Select WO 5 or WO 7. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0324	**Error message** Incorrect tool for machining. Select WO 1 or WO 3 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools..
603-0325	**Error message** Incorrect tool for machining. Select WO 1 or WO 2 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0326	**Error message** Incorrect tool for machining. Select WO 2 or WO 3 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0327	**Error message** Incorrect tool for machining. Select WO 3 or WO 4 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.

Error number	Description
603-0328	**Error message** Incorrect tool for machining. Select WO 4 or WO 5 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0329	**Error message** Incorrect tool for machining. Select WO 5 or WO 6 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-032A	**Error message** Incorrect tool for machining. Select WO 6 or WO 7 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-032B	**Error message** Incorrect tool for machining. Select WO 7 or WO 8 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-032C	**Error message** Incorrect tool for machining. Select WO 8 or WO 1 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.

Error number	Description
603-032D	**Error message** Incorrect tool for machining. Select WO 2, WO 3, WO 5 or WO 6 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools. CAUTION: The selection of the tool determines whether the contour is machined from the left or right!
603-032E	**Error message** Incorrect tool for machining. Select WO 1, WO 2, WO 6 or WO 7 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools. {0>ACHTUNG:<}0{>CAUTION:<0} The selection of the tool determines whether the contour is machined from the left or from the right!
603-032F	**Error message** Incorrect tool for machining. Select WO 1, WO 8, WO 3 or WO 4 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools. ACHTUNG: The selection of the tool determines whether the contour is machined from the left or right!
603-0330	**Error message** Incorrect tool for machining. Select WO 4, WO 5, WO 7 or WO 8 %1 **Cause of error** An unsuitable tool was chosen for the intended operation. **Error correction** Select one of the proposed tools CAUTION: The selection of the tool decides whether cutting is on the left or right of the contour!

Error number	Description
603-0331	**Error message** Incorrect tool for machining. Select WO 1, WO 3, WO 5 or WO 7 %1 **Cause of error** An unsuitable tool was chosen for the intended operation. **Error correction** Select one of the proposed tools CAUTION: The selection of the tool decides whether cutting is on the left or right of the contour!
603-0332	**Error message** Incorrect tool for machining. Select WO 2, WO 4, WO 6 or WO 8 %1 **Cause of error** An unsuitable tool was chosen for the intended operation. **Error correction** Select one of the proposed tools CAUTION: The selection of the tool decides whether cutting is on the left or right of the contour!
603-0333	**Error message** The machining contour exceeds the center of turning Modify the corresponding coordinates! %1 **Cause of error** The machining contour corners were programmed past the turning center **Error correction** Enter the coordinates so they all lie on one side of the center of rotation
603-0334	**Error message** No surface for machining Cycle has nothing to process! %1 **Cause of error** The cycle cannot calculate a machining surface because the contour only consists of one paraxial segment. **Error correction** Edit the entered coordinates

Error number	Description
603-0335	**Error message** Tool position not permissible! Position in front of, or over the machining area! %1 **Cause of error** The cycle cannot move the tool from its given position to the starting point of machining without danger of collision. **Error correction** Position the tool in front of or over the machining area
603-0336	**Error message** RAM data memory is full! Contact Service! %1 **Cause of error** Insufficient memory for contour calculation **Error correction** This problem can only be fixed by service personnel.
603-0337	**Error message** Incorrect tool for turning. Select a lathe tool. %1 **Cause of error** An unsuitable tool was chosen for the intended operation. **Error correction** Select one of the proposed tools CAUTION: The selection of the tool decides whether cutting is on the left or right of the contour!
603-0338	**Error message** Incorrect tool for cutting off. Select a recess. tool. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools. CAUTION: The selection of the tool determines whether the contour is machined from the left or right!

Error number	Description
603-0339	**Error message** Incorrect tool for machining. Select WO 1, WO 3 or WO 2. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools..
603-033A	**Error message** Incorrect tool for machining. Select WO 1, WO 3 or WO 4. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-033B	**Error message** Incorrect tool for machining. Select WO 2, WO 8 or WO 1. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select on of the proposed tools.
603-033C	**Error message** Incorrect tool for machining. Select WO 2, WO 8 or WO 7. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select on of the proposed tools.
603-033D	**Error message** Incorrect tool for machining. Select WO 4, WO 6 or WO 3. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select on of the proposed tools.

Error number	Description
603-033E	**Error message** Incorrect tool for machining. Select WO 4, WO 6 or WO 5. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select on of the proposed tools.
603-033F	**Error message** Incorrect tool for machining. Select WO 5, WO 7 or WO 6. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0340	**Error message** Incorrect tool for machining. Select WO 5, WO 7 or WO8. %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0341	**Error message** Incorrect tool for machining. Select WO 1, WO 2, WO 3, WO 4 or WO 5 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0342	**Error message** Incorrect tool for machining. Select WO 3, WO 4 or WO 5 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.

Error number	Description
603-0343	**Error message** Incorrect tool for machining. Select WO 3, WO 4, WO 5, WO 6 or WO 7 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0344	**Error message** Incorrect tool for machining. Select WO 5, WO 6 or WO 7 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0345	**Error message** Incorrect tool for machining. Select WO 5, WO 6, WO 7, WO 8 or WO 1 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0346	**Error message** Incorrect tool for machining. Select WO 7, WO 8 or WO 1 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0347	**Error message** Incorrect tool for machining. Select WO 7, WO 8, WO 1, WO 2 or WO 3 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.

Error number	Description
603-0348	**Error message** Incorrect tool for machining. Select WO 1, WO 2 or WO 3 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0355	**Error message** Incorrect tool for machining. Select WO 5, WO 6, WO 7, WO 8 or WO 1 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0356	**Error message** Incorrect tool for machining. Select WO 7, WO 8 or WO 1 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0357	**Error message** Incorrect tool for machining. Select WO 7, WO 8, WO 1, WO 2 or WO 3 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0358	**Error message** Incorrect tool for machining. Select WO 1, WO 2 or WO 3 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.

Error number	Description
603-0359	**Error message** Incorrect tool for machining. Select WO 1, WO 2, WO 3, WO 4 or WO 5 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-035A	**Error message** Incorrect tool for machining. Select WO 3, WO 4 or WO 5 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-035B	**Error message** Incorrect tool for machining. Select WO 3, WO 4, WO 5, WO 6 or WO 7 %1 **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-035C	**Error message** Incorrect tool for machining. Select WO 5, WO 6 or WO 7 %1 The main cutting direction does not fit the contour. **Cause of error** An unsuitable tool was selected for the next operation. **Error correction** Select one of the proposed tools.
603-0360	**Error message** Invalid tool orientation for transverse machining (roughing) **Cause of error** In the facing cycle (transverse machining), tool orientations 2 and 6 are not possible for roughing **Error correction**

Error number	Description
603-0361	**Error message** Invalid tool orient. for longitudinal machining (roughing) **Cause of error** In longitudinal machining, tool orientations 4 and 8 are not possible for roughing **Error correction**
603-0366	**Error message** The cycle cannot completely create the programmed contour; residual material will remain. %1 **Cause of error** Aufgrund von z. B. der Schneidengeometrie und/oder dem Anstellwinkel zwischen Werkzeug und Werkstück kann der Zyklus Teile der programmierten Kontur nicht erreichen. **Error correction** Prüfen Sie die aktuelle Bearbeitungssituation und beurteilen Sie, ob - die zu bearbeitende Kontur - das gewählte Werkzeug - die gewählte Anstellung - der programmierte Zyklus der geforderten Bearbeitung entspricht bzw. korrigieren Sie diese. Prüfen Sie, ob ein evtl. nachfolgender (Schlicht-)Zyklus das Restmaterial ohne Überlastung des Werkzeugs bearbeiten kann und korrigieren Sie auch hier gegebenenfalls die oben genannten Parameter. Falls das verbleibende Restmaterial für ihre Bearbeitungssituation akzeptabel ist, können Sie diese Meldung ignorieren.
605-024E	**Error message** Programmed cutting depth too small %1 **Cause of error** The programmed value is too small. **Error correction** Check the NC program
605-0278	**Error message** Feed axis is already braked in the thread %1 **Cause of error** The run-out length P in the thread cycle G31 is less than the cutting width SB of the tool being used. **Error correction** Enter a run-out length P at least as large as the cutting width SB of the tool, or select a tool with a smaller cutting width SB.

Error number	Description
605-0279	**Error message** Feed axis is still accelerated in the thread %1 **Cause of error** The run-in distance of the feed axis is too small. The tool cannot achieve the speed necessary in order to cut the thread correctly. **Error correction** Increase the run-in length B
605-027A	**Error message** Determined run-in length is too short %1 **Cause of error** The space automatically determined from the recess or undercut is too short for the run-in length. **Error correction** Program a run-in length, or increase the recess or undercut
605-027B	**Error message** Determined run-out length is too short %1 **Cause of error** The space automatically determined from the recess or undercut is too short for the run-out length. **Error correction** Program a run-out length, or increase the recess or undercut
605-032C	**Error message** "TOOL_P" configured without tool magazine **Cause of error** The path to a pocket table was given in the machine configuration although no tool magazine is configured. Access by the PLC run-time system to the pocket table is enabled through the symbolic name "TOOL_P" only for tool magazines. **Error correction** Adapt the machine configuration: - Delete the "TOOL_P" path for the pocket table if no tool magazines are configured.

Error number	Description
605-032D	**Error message** Tool magazine configured without path for "TOOL_P" **Cause of error** Tool magazines were configured without the TOOL_P path for the pocket table. **Error correction** Adapt the machine configuration: - Enter the symbolic name "TOOL_P" for the pocket table in CfgTablePath - Enter the path to the pocket table under "TOOL_P"
605-0342	**Error message** Active tool pocket unknown **Cause of error** The tool change was canceled. **Error correction** - Correct any pending error messages and repeat - Inform your service agency
605-0344	**Error message** Protective zone monitoring not allowed in axis system with B axis **Cause of error** The protection zone monitor in the axis system does monitor any position changes of the tool by rotary axes (e.g. the B axis). **Error correction** - Activation of the expanded protection zone monitor is required (MP_enhancedProtZone = 2: Machine base system) - In addition, for machines with rear-face machining, adjust-ments in the kinematics is required - Contact the machine tool builder - Inform your service agency
605-0358	**Error message** Starting block of channel <%2> cannot be reached because other channels are already waiting at the sync. point %1 **Cause of error** The starting blocks were set on the individual channels in such a manner that, because of sync. points, they cannot be reached. **Error correction** Cancel the mid-program startup, and set the starting block before the sync. point.

Error number	Description
605-0359	**Error message** Program run blocked due to a serious error (e.g., configuration data or table) %1 **Cause of error** The configuration data or tables are not complete. Reliable program execution cannot be performed. See the LogFile.log for more information. **Error correction** Correct the configuration data and complete the tables.
606-0062	**Error message** Calculation of the approach path not possible %1 **Cause of error** All calculated approaching paths damage the workpiece. **Error correction** Select another tool position
606-02E2	**Error message** Area to be machined was not provided %1 **Cause of error** The area to be machine has not been provided. **Error correction** - Check the NC program - Contact your service agency if the error occurs during program generation with TURN PLUS
606-02E3	**Error message** A negative safety clearance is not allowed %1 **Cause of error** Global machining parameters are incorrect or safety clearance is set to an invalid value **Error correction** Revise the global machining parameters or set the safety clearance before the cycle call
606-0343	**Error message** No valid cutting depth was defined; cycle is working with 2/3 of the tool’s max. cutting depth %1 **Cause of error** The value of the G function’s P parameter is less than or equal to 0. **Error correction** Reset the P parameter of the NC block

Error number	Description
606-0345	**Error message** Internal memory of calculating the cuts is full %1 **Cause of error** The cycle needs too much memory space to save all cut lines. Example: 50 mm oversize with 0.003 mm cutting depth. **Error correction** Enter a larger cutting depth
606-0385	**Error message** Too many stepping angles for simultaneous turning cycle **Cause of error** Calculation of the simultaneous turning cycle takes too long because the resolution of the angle range is too high. The resolution was reduced automatically. **Error correction**
606-0386	**Error message** Not a valid machining contour for simultaneous turning cycle No valid tool contour for simultaneous turning cycle **Cause of error** - The cycle has received an invalid contour. - Internal software error. **Error correction** Check the contour: workpiece blank, tool holder, and machining contour must be closed. All elements must be greater than zero. - Inform your service agency.
606-0387	**Error message** Contour elements not properly marked (simultaneous turning) **Cause of error** The contour elements of the center point contour are not correctly marked for the simultaneous turning cycle. **Error correction** The following conditions must be fulfilled: - An open space of the contour must be marked as "CYC". - The starting element of the space must be the first element of the contour.

Error number	Description
606-0388	**Error message** Internal error in the simultaneous turning cycle **Cause of error** Due to an internal error, the simultaneous turning cycle could not be executed. **Error correction** - See "INTERNAL INFO" for more information. - Generate the service files and inform your service agency.
606-0389	**Error message** The contour cannot be completely machined **Cause of error** With the selected parameters, the simultaneous turning cycle cannot completely finished the programmed contour without collision. **Error correction** Inasmuch as is technically meaningful, adapt the range of the inclination angle in the cycle and, if required, select another tool for machining.
606-038B	**Error message** Simultaneous turning: Tool is too far "behind" input contour **Cause of error** Correct approach and departure behavior is not possible if the tool along the Z axis is behind the workpiece blank or the machining contour. **Error correction** Position the tool farther away from the chuck (along Z+) in order to enable correct approach and departure behavior.
606-038C	**Error message** The tilting motion results in a collision with the tool **Cause of error** The cycle could not rotate the tool from the initial inclination to the start inclination calculated by the cycle, or from the end inclination back to the initial inclination. **Error correction** Position the tool outside the collision range, farther away from the workpiece.
606-038D	**Error message** Simultaneous turning: Tool tip radius must not be null **Cause of error** The simultaneous turning cycle requires a tool with a cutter radius greater than zero. **Error correction** Select another tool for the operation

Error number	Description
606-038E	**Error message** Tool position not defined **Cause of error** For open contours, the tool position must be programmed for the simultaneous turning cycle. **Error correction** Enter whether the tool should move to the right or left of the contour.
606-0391	**Error message** Collision-free approach/departure could not be calculated **Cause of error** No collision-free approach/departure paths could be calculated for the given turning contour and position of the tool. **Error correction** It may be that the contour does not allow collision-free approach/departure paths. Check the contour, and correct the position of the tool if necessary.
606-0392	**Error message** Desired inclination angles can't be reached without collision Cannot reach desired angle of incidence at beginning of contour %1 **Cause of error** The cycle could not be run because the programmed inclination angles could cause collisions. **Error correction** Change the NC program - adjust the inclination angle accordingly
606-0394	**Error message** Clearance angle + cutting edge angle must be less than 180 %1 **Cause of error** Clearance angle + cutting edge must not be more than 180°. **Error correction** - Decrease the clearance angle or - Select a different tool

Error number	Description
606-0396	**Error message** Cycle changes angle of incidence at starting point %1 **Cause of error** An inclination angle was defined at the beginning of machining, together with the option "do not approach". **Error correction** - Select a different type of approach or - Approach the desired inclination angle manually
606-0397	**Error message** Aux. contour of tool head faulty, collision monitoring not possible %1 **Cause of error** The description of the tool head does not include a closed envelope. **Error correction** Adapt or correct the description of the tool head.
606-0398	**Error message** Wear comp. violates holder oversize %1 **Cause of error** The wear compensations of the tool are greater than the selected holder oversize **Error correction** Check the wear compensations and the holder oversize; you might need to choose a different tool
606-0399	**Error message** Maximum infeed exceeded **Cause of error** The desired infeed must be chosen so that it is less than the maximum infeed. **Error correction** Reduce the desired infeed or increase the maximum infeed.
606-039A	**Error message** Initial tool angle not in permitted inclination range **Cause of error** The pre-positioned tool angle violates the minimum or maximum inclination angle. **Error correction** Either pre-position the tool angle within the permitted angle range or expand the angle range correspondingly

Error number	Description
606-039B	**Error message** %2 Residual material will remain %1 **Cause of error** The target contour cannot be completely machined; residual material will remain. This can be due to various reasons, such as the tool cannot reach the respective area due to geometrical reasons, the prescribed inclination angle range makes it impossible to reach the residual material, etc. **Error correction** Residual material must be considered for the subsequent machining operations.
606-039C	**Error message** The cycle cannot machine the indicated target contour **Cause of error** The cycle cannot use the defined input parameters and the selected tool to machine the target contour. **Error correction** Adapt the corresponding input parameters or adapt the target contour or select an appropriate tool.
606-039D	**Error message** Maximum infeed of tool exceeded **Cause of error** The desired infeed exceeds 2/3 of the cutting edge length of the tool. Cutting lines will be adapted if necessary. **Error correction** Reduce the desired infeed or select an appropriate tool
606-039F	**Error message** The infeed does not match the length of the cutting edge **Cause of error** Possible causes: - The current infeed is greater than the maximum cutting depth - The maximum cutting depth is greater than 2/3 of the current length of the cutting edge **Error correction** Reduce the (maximum) infeed or use an appropriate tool

Error number	Description
606-03A0	**Error message** The safety clearance is too small %1 **Cause of error** The cycle requires a greater safety clearance in order to work without collision. **Error correction** Program a greater safety clearance.
60F-0033	**Error message** Finished part no longer located within current workpiece. %1 **Cause of error** An area of the finished part, e.g. an end point, lies outside of the defined workpiece blank. This note has no effect on the program run. **Error correction**
612-0001	**Error message** System warning: %1 **Cause of error** System warning. A recoverable internal error has occurred. The process will NOT be canceled. **Error correction**
612-0002	**Error message** System error: %1 **Cause of error** A fatal error has occurred. The process will be canceled. **Error correction**
620-004D	**Error message** Tool ID %1 not found Tool not found (%1) **Cause of error** A tool not included in the tool table has been entered in the turret or magazine assignment. **Error correction** Remove the tool from the turret/magazine assignment, or add the tool to the tool table.

Error number	Description
621-003F	**Error message** Helix diameter is greater than twice the cutter diameter %1 **Cause of error** With the selected cutter diameter, material remains standing in the middle of the plunging helix. **Error correction** Program a smaller “Diameter of helix”
621-0040	**Error message** Pilot holes were calculated for a different contour %1 **Cause of error** For the calculation of the pilot hole positions, a different contour was programmed than the one in the current milling cycle. **Error correction** Change the position marker of the pilot holes or exchange the contour to be machined
621-0041	**Error message** Pilot holes were calculated for a different trochoid width %1 **Cause of error** For the calculation of the pilot hole positions, a different trochoid width was programmed than the one in the current milling cycle. **Error correction** Correct the trochoid width.
621-0042	**Error message** Pilot holes were calculated with a different oversize %1 **Cause of error** For the calculation of the pilot hole positions, a different oversize was programmed than the one in the current milling cycle. **Error correction** Correct the oversize

Error number	Description
621-0043	**Error message** No corners for trochoidal milling present %1 **Cause of error** For slots and circular pockets, there are no corners to be milled trochoidally. **Error correction** Program the machining operation as “Complete” or “W/o corner machining”
621-0044	**Error message** Pilot holes on the other side of the contour %1 **Cause of error** For the calculation of the pilot hole positions, a different tool position was programmed than the one in the current milling cycle. **Error correction** Correct the tool position
621-0045	**Error message** Tool diameter is too large %1 **Cause of error** The tool diameter must be smaller than the plunging length or the diameter of the helix. **Error correction** Use a suitable tool
621-0047	**Error message** Cutter diameter must be less than the trochoid width %1 **Cause of error** With the current cutter diameter and the programmed trochoid width, no trochoidal paths can be calculated. **Error correction** Select a cutter with smaller diameter, or program a larger trochoid width
621-0048	**Error message** Radius for return greater than half the trochoid width %1 **Cause of error** With the programmed radius, the path for return lies outside of the trochoidal path. **Error correction** Program a smaller radius for return or a larger trochoid width

Error number	Description
621-0049	**Error message** Trochoid width smaller than plunging helix radius %1 **Cause of error** The plunging paths lie partially outside of the trochoidal path. **Error correction** Program a smaller plunging helix or larger trochoid width
621-004A	**Error message** Pilot holes not present %1 **Cause of error** No position markers were programmed under the specified position marker. **Error correction** Correct the position marker
621-004B	**Error message** Depth of the pilot hole less than the milling depth %1 **Cause of error** The drilling depth must not be less than the milling depth. **Error correction** Correct the drilling or milling depth
621-004C	**Error message** No trochoid width programmed %1 **Cause of error** Without a specified trochoid width, the cycle cannot calculate any paths. **Error correction** Program the trochoid width
621-004D	**Error message** Programmed oversize too large %1 **Cause of error** The oversize is so large that inside machining of the contour is no longer possible. **Error correction** Correct the oversize

Error number	Description
621-004E	**Error message** Programmed trochoid width too large %1 **Cause of error** The trochoid width must be less than the slot width or rectangle width when the oversize values are taken into account. **Error correction** Correct the trochoid width.
663-04EA	**Error message** Error when loading a dialog: %1 **Cause of error** The dialog box could not be opened because of a faulty or missing dialog description by the machine manufacturer. **Error correction** Inform your service agency Inform your machine tool builder
900-0BB8	**Error message** File '%1' not found **Cause of error** The given file path does not refer to a graphic file. **Error correction** Select another graphic file.
900-0BB9	**Error message** Failed to send internal message **Cause of error** Error in the internal system communication. **Error correction** Inform your service agency.
900-0BBA	**Error message** Unable to open configuration server queue **Cause of error** Error in the internal system communication. **Error correction** Inform your service agency.
900-0BBB	**Error message** Unable to read configuration data '%1' **Cause of error** Error in the internal system communication. **Error correction** Inform your service agency.

Error number	Description
900-0BBC	**Error message** Configuration data '%1' could not be written **Cause of error** Error in the internal system communication. **Error correction** Inform your service agency.
900-0BBD	**Error message** Internal error! **Cause of error** Internal GRED software error. **Error correction** Inform your service agency.
900-0BBE	**Error message** Internal error: %1 **Cause of error** Internal GRED software error. **Error correction** Inform your service agency.
900-0BBF	**Error message** Invalid data in graphic file: %1 **Cause of error** The existing data cannot be read as graphic data. **Error correction** - Correct/remove and save the corresponding data, or - Delete the graphic file and make a new one
900-0BC0	**Error message** No graphic file: %1 **Cause of error** The specified file cannot be read as a graphic file. **Error correction** Select another graphic file.
900-0BC1	**Error message** Missing object '%1' **Cause of error** Missing object in the selected graphic file. **Error correction** - Add the corresponding object and save, or - Delete the graphic file and make a new one

Error number	Description
900-0BC2	**Error message** Object '%1' is incomplete **Cause of error** Missing object elements in the selected graphic file. **Error correction** - Correct the corresponding object and save, or - Delete the graphic file and make a new one
900-0BC3	**Error message** Object '%1' already exists **Cause of error** Displayed object already exists in the selected graphic file. **Error correction** - Remove the redundant object and save, or - Delete the graphic file and make a new one
900-0BC4	**Error message** Working plan '%1' is incomplete **Cause of error** Steps are still missing in the displayed working plan. **Error correction** - Add the missing work steps and save, or - Remove all entries in the working plan
900-0BC5	**Error message** Geometry object list is incomplete **Cause of error** More geometry objects are needed to fully describe the workpiece form. **Error correction** - Add the missing geometry objects and save, or - Delete the graphic file and make a new one
900-0BC6	**Error message** Redundant object '%1' present **Cause of error** A redundant object is in the selected graphic file and will no longer be used. **Error correction** Remove the redundant object manually and save: - Remove the object in the graphic file and save the file, or - Resave the loaded graphic file

Error number	Description
900-0BC7	**Error message** Environment variable '%1' not defined **Cause of error** Internal GRED NC program template error. **Error correction** Inform your service agency.
900-0BC8	**Error message** Control structure variable '%1' not defined **Cause of error** Internal GRED NC program template error. **Error correction** Inform your service agency.
900-0BC9	**Error message** Maximum nesting depth of control structures has been reached **Cause of error** Internal GRED NC program template error. **Error correction** Inform your service agency.
900-0BCA	**Error message** Faulty composition of control structure **Cause of error** Internal GRED NC program template error. **Error correction** Inform your service agency.
900-0BCB	**Error message** Unknown control structure '%1' **Cause of error** Internal GRED NC program template error. **Error correction** Inform your service agency.
900-0BCC	**Error message** Invalid value '%1' in object '%2' **Cause of error** There is an incorrect attribute value in the current object. **Error correction** Use a text editor to check and correct the attribute value in the object and to save it again to the corresponding graphic file.

Error number	Description
900-0BCD	**Error message** Tool data incomplete: '%1' **Cause of error** Required data on the tool are missing. **Error correction** Add the required tool data to the workpiece table and save it.
900-0BCE	**Error message** Error when opening the tool table '%1' **Cause of error** The corresponding tool table cannot be opened: - Tool table is not missing in the given file path - Tool table has an invalid format - Tool table is inconsistent **Error correction** Inform your service agency.
900-0BCF	**Error message** New parameters of input form '%1' inserted **Cause of error** Some required parameters are missing in the form. **Error correction** The system has already made the correction. Please check the result!
900-0BD0	**Error message** Old parameters of input form '%1' removed **Cause of error** There are too many parameters in the current form. **Error correction** The system has already made the correction. Please check the result!
900-0BD1	**Error message** Variable name '%1' already exists **Cause of error** Internal GRED NC program template error. **Error correction** Inform your service agency.

Error number	Description
900-0BD2	**Error message** Graphic file '%1' cannot be opened **Cause of error** An error occurred when the graphic file was opened. **Error correction** Ensure that the graphic file exists, that the given path is correct, and that the file is in a readable format.
900-0BD3	**Error message** Graphic file '%1' is too large **Cause of error** The graphic file does not fit into the space provided. **Error correction** Ensure that the graphic file fulfills the required dimensions.
900-0BD4	**Error message** Error when writing the tool data '%1' **Cause of error** The tool cannot be saved to the table. **Error correction** Check the corresponding tool data.
900-0BD5	**Error message** Object '%1' is faulty **Cause of error** Incorrect object elements in the selected graphic file. **Error correction** Correct the corresponding object and save.
900-0BD6	**Error message** Parameter '%1' not defined **Cause of error** The given parameter is not fully defined in the configuration. **Error correction** Complete the definition of the configuration and save.
900-0BD7	**Error message** Environment variable '%1' not initialized **Cause of error** Internal GRED NC program template error. **Error correction** Inform your service agency.

Error number	Description
900-0BD8	**Error message** Error when reading the tool data '%1' **Cause of error** Required data on the tool are missing. **Error correction** Add the corresponding tool data.
900-0BD9	**Error message** Configuration data are incomplete **Cause of error** The given references/information regarding configuration data are incomplete. **Error correction** Complete the configuration data.
900-0BDA	**Error message** Parameter '%1' not allowed in this formula! **Cause of error** The input form lists an illegal parameter that cannot be evaluated. **Error correction** Remove the corresponding form parameter and replace it with another.
900-0BDB	**Error message** Configuration object '%1' is faulty **Cause of error** Configuration object has incorrect or incomplete data. **Error correction** Correct the configuration object and save.
903-0001	**Error message** Starting position of noncircular contour not in workpiece system Starting position of noncircular contour not in workpiece system **Cause of error** - Axis-value programming is active **Error correction** - Edit the program

Error number	Description
903-0002	**Error message** Polar starting position is programmed incrementally Polar starting position is programmed incrementally **Cause of error** The radius or angle of the polar-programmed starting position is programmed incrementally **Error correction** Edit the program or the cycle
903-0003	**Error message** Starting position is programmed incrementally Starting position is programmed incrementally **Cause of error** Polar programmed starting position is programmed incrementally **Error correction** Edit the program or the cycle
903-0004	**Error message** Relative programming of the reciprocation starting position is not allowed Relative programming of the reciprocation starting position is not allowed **Cause of error** Starting position of the reciprocation is programmed in relative values **Error correction** Program the starting position with absolute values
903-0005	**Error message** Absolute programming of the relief vector is not allowed Absolute programming of the relief vector is not allowed **Cause of error** - Relief vector was programmed absolutely instead of incrementally **Error correction** - Program the relief vector incrementally

Error number	Description
903-0006	**Error message** Relative programming of the infeed starting position is not allowed! Relative programming of the infeed starting position is not allowed! **Cause of error** Starting position of the infeed is programmed in relative values **Error correction** Program the starting position with absolute values
903-0007	**Error message** Programmed axis is not a grinding axis Programmed axis is not a grinding axis **Cause of error** Wrong axis selected for grinding **Error correction** Program a grinding axis
903-0008	**Error message** Programmed value is not interpreted as a coordinate Programmed value is not interpreted as a coordinate **Cause of error** Presumably a system error **Error correction** Inform your service agency
903-0009	**Error message** Incremental programming of the starting position is not allowed Incremental programming of the starting position is not allowed **Cause of error** Starting position is programmed incrementally **Error correction** Program the starting position with absolute values
903-000A	**Error message** Reciprocation over zero length is not allowed Reciprocation over zero length is not allowed **Cause of error** **Error correction** - Edit the program

Error number	Description
905-2711	**Error message** Pre-position [Ls] **Cause of error** **Error correction**
905-2712	**Error message** Thread type (0=ext. / 1=int.) **Cause of error** **Error correction**
905-2713	**Error message** Thread pitch **Cause of error** **Error correction**
905-2714	**Error message** Thread depth **Cause of error** **Error correction**
905-2715	**Error message** Amount of infeed **Cause of error** **Error correction**
905-2716	**Error message** Remaining cut division (0 = Yes) **Cause of error** **Error correction**
905-2717	**Error message** Number of dry runs **Cause of error** **Error correction**
905-2718	**Error message** Run-out length at end of thread **Cause of error** **Error correction**

Error number	Description
905-2719	**Error message** Starting angle **Cause of error** **Error correction**
905-271A	**Error message** Rotational speed (rpm) **Cause of error** **Error correction**
905-271B	**Error message** Peripheral speed (m/min) **Cause of error** **Error correction**
905-271C	**Error message** Workpiece rotation speed (1/min) **Cause of error** **Error correction**
905-271D	**Error message** Taper angle (>0 = ascending) **Cause of error** **Error correction**
905-271F	**Error message** Stndrd thread (0=none, 1=ISO,..) **Cause of error** **Error correction**
905-2720	**Error message** Nominal diameter **Cause of error** **Error correction**
905-2721	**Error message** Thread 0=righthand,1=lefthand **Cause of error** **Error correction**

Error number	Description
905-2722	**Error message** Return **Cause of error** **Error correction**
905-2723	**Error message** Run-in length **Cause of error** **Error correction**
905-2724	**Error message** Run-in speed **Cause of error** **Error correction**
905-2725	**Error message** Length of cut in depth **Cause of error** **Error correction**
905-2726	**Error message** Traversing speed **Cause of error** **Error correction**
905-2727	**Error message** Run-out length **Cause of error** **Error correction**
905-2728	**Error message** Speed on run-out **Cause of error** **Error correction**
905-2729	**Error message** Target position X **Cause of error** **Error correction**

Error number	Description
905-272A	**Error message** Target position Z **Cause of error** **Error correction**
905-272B	**Error message** Reciprocation feed rate [F] **Cause of error** **Error correction**
905-272C	**Error message** Infeed rate [D] **Cause of error** **Error correction**
905-272D	**Error message** Number of spark-out strokes [H] **Cause of error** **Error correction**
905-2737	**Error message** Tool number (T0 - T9) **Cause of error** **Error correction**
905-2738	**Error message** Error number **Cause of error** **Error correction**
905-2739	**Error message** Error consequence **Cause of error** **Error correction**
905-273A	**Error message** Error level **Cause of error** **Error correction**

Error number	Description
905-273B	**Error message** Wheel location **Cause of error** **Error correction**
905-273C	**Error message** Dresser location **Cause of error** **Error correction**
905-273D	**Error message** Wheel edge **Cause of error** **Error correction**
905-273E	**Error message** Place – Bit **Cause of error** **Error correction**
905-273F	**Error message** Type of machining **Cause of error** **Error correction**
905-2740	**Error message** Command number **Cause of error** **Error correction**
905-2741	**Error message** Dresser number **Cause of error** **Error correction**
905-2742	**Error message** Value 1 **Cause of error** **Error correction**

Error number	Description
905-2743	**Error message** Value 2 **Cause of error** **Error correction**
905-2744	**Error message** Value 3 **Cause of error** **Error correction**
905-2745	**Error message** Calibration **Cause of error** **Error correction**
905-2746	**Error message** Encoder **Cause of error** **Error correction**
905-2747	**Error message** Linear measurement **Cause of error** **Error correction**
905-2748	**Error message** Diameter **Cause of error** **Error correction**
905-2749	**Error message** Delete Z **Cause of error** **Error correction**
905-274A	**Error message** Delete X **Cause of error** **Error correction**

Error number	Description
905-274B	**Error message** Define the wheel shape **Cause of error** **Error correction**
905-274C	**Error message** Teach in width **Cause of error** **Error correction**
905-274D	**Error message** Define the tool **Cause of error** **Error correction**
905-274E	**Error message** Continue **Cause of error** **Error correction**
905-274F	**Error message** Question mark Question mark **Cause of error** **Error correction**
905-2750	**Error message** Teach in Z Teach in Z **Cause of error** **Error correction**
905-2751	**Error message** Teach in X Teach in X **Cause of error** **Error correction**
905-2752	**Error message** Delete the dresser Delete the dresser **Cause of error** **Error correction**

Error number	Description
905-2753	**Error message** Spindle dresser **Cause of error** **Error correction**
905-2754	**Error message** Dress plate **Cause of error** **Error correction**
905-2755	**Error message** Diamond dresser **Cause of error** **Error correction**
905-2756	**Error message** Dresser alignment **Cause of error** **Error correction**
905-2757	**Error message** Teach in dresser position **Cause of error** **Error correction**
905-2758	**Error message** New dresser **Cause of error** **Error correction**
905-2759	**Error message** V V **Cause of error** **Error correction**
905-275A	**Error message** Transmission ratio **Cause of error** **Error correction**

Error number	Description
905-275B	**Error message** Initialize Initialize **Cause of error** **Error correction**
905-275C	**Error message** Calculate Calculate **Cause of error** **Error correction**
905-275D	**Error message** Reload the data **Cause of error** **Error correction**
905-275E	**Error message** Settings Settings **Cause of error** **Error correction**
905-275F	**Error message** Outer side Outer side **Cause of error** **Error correction**
905-2760	**Error message** Inner side Inner side **Cause of error** **Error correction**
905-2761	**Error message** Search criteria **Cause of error** **Error correction**

Error number	Description
905-2762	**Error message** Confirm the data **Cause of error** **Error correction**
905-2763	**Error message** Next Next **Cause of error** **Error correction**
905-2764	**Error message** Previous Previous **Cause of error** **Error correction**
905-2765	**Error message** Next identical **Cause of error** **Error correction**
905-2766	**Error message** Previous identical **Cause of error** **Error correction**
905-2767	**Error message** Tool information **Cause of error** **Error correction**
905-2768	**Error message** Internal grinding **Cause of error** **Error correction**
905-2769	**Error message** External grinding **Cause of error** **Error correction**

Error number	Description
905-276A	**Error message** General wheel data **Cause of error** **Error correction**
905-276B	**Error message** Face plate **Cause of error** **Error correction**
905-276C	**Error message** Angular wheel **Cause of error** **Error correction**
905-276D	**Error message** Straight wheel **Cause of error** **Error correction**
905-276E	**Error message** End ? **Cause of error** **Error correction**
905-276F	**Error message** Starting position X **Cause of error** **Error correction**
905-2770	**Error message** Starting position Z **Cause of error** **Error correction**
905-2771	**Error message** End position X **Cause of error** **Error correction**

Error number	Description
905-2772	**Error message** End position Z **Cause of error** **Error correction**
905-2773	**Error message** X value for swing position 1 **Cause of error** **Error correction**
905-2774	**Error message** Z value for swing position 1 **Cause of error** **Error correction**
905-2775	**Error message** X value for swing position 2 **Cause of error** **Error correction**
905-2776	**Error message** Z value for swing position 2 **Cause of error** **Error correction**
905-2777	**Error message** Reciprocation feed rate [F1] **Cause of error** **Error correction**
905-2778	**Error message** Reciprocation feed rate [F2] **Cause of error** **Error correction**
905-2779	**Error message** Infeed rate [F] **Cause of error** **Error correction**

Error number	Description
905-277A	**Error message** Amount of infeed [D] **Cause of error** **Error correction**
905-277B	**Error message** Search path of the probe **Cause of error** **Error correction**
905-277C	**Error message** Offset [L] Offset [L] **Cause of error** **Error correction**
905-277D	**Error message** Dwell time [H] **Cause of error** **Error correction**
905-277E	**Error message** Dwell time [H] at swing pos. 1 **Cause of error** **Error correction**
905-277F	**Error message** Dwell time [H2] at swing pos. 2 **Cause of error** **Error correction**
905-2780	**Error message** Number of spark-out strokes [N] **Cause of error** **Error correction**
905-2781	**Error message** Selection of override switch [O] **Cause of error** **Error correction**

Error number	Description
905-2782	**Error message** Retraction amount [A] **Cause of error** **Error correction**
905-2783	**Error message** Absolute (=0) or relative (=1) **Cause of error** **Error correction**
905-2784	**Error message** Referenced to axis 1=yes, 0=no **Cause of error** **Error correction**
905-2785	**Error message** Infeed in pos. 1, 2 or both **Cause of error** **Error correction**
905-2786	**Error message** Faulty behavior with undersize **Cause of error** **Error correction**
905-2787	**Error message** Faulty behavior with oversize **Cause of error** **Error correction**
905-2788	**Error message** Shift of end position [K] **Cause of error** **Error correction**
905-2789	**Error message** Sensor already active at start **Cause of error** **Error correction**

Error number	Description
905-278A	**Error message** Sensor has not responded **Cause of error** **Error correction**
905-278B	**Error message** Probe already active at start **Cause of error** **Error correction**
905-278C	**Error message** Probe has not responded **Cause of error** **Error correction**
905-278D	**Error message** Type of status information 0-4 **Cause of error** **Error correction**
905-278E	**Error message** Type of movement 0-2 **Cause of error** **Error correction**
905-278F	**Error message** Mode of execution 0-1 **Cause of error** **Error correction**
905-2790	**Error message** Type of movement 0-8 **Cause of error** **Error correction**
905-2791	**Error message** Type of movement 0-3 **Cause of error** **Error correction**

Error number	Description
905-2792	**Error message** Type of event **Cause of error** **Error correction**
905-2793	**Error message** Type of reaction **Cause of error** **Error correction**
905-2794	**Error message** Signal for OK **Cause of error** **Error correction**
905-2795	**Error message** Reaction in event test 1 **Cause of error** **Error correction**
905-2796	**Error message** Reaction in event test 2 **Cause of error** **Error correction**
905-2797	**Error message** Reaction in event test 3 **Cause of error** **Error correction**
905-2798	**Error message** Dressing amount [D] **Cause of error** **Error correction**
905-2799	**Error message** Dressing feed rate [F] **Cause of error** **Error correction**

Error number	Description
905-279A	**Error message** Number of repetitions [E] **Cause of error** **Error correction**
905-279B	**Error message** Number of idle strokes [H] **Cause of error** **Error correction**
905-279C	**Error message** Angle of infeed [Q] **Cause of error** **Error correction**
905-279D	**Error message** Dressing after number of pcs. **Cause of error** **Error correction**
905-279E	**Error message** Diamond number for outer side **Cause of error** **Error correction**
905-279F	**Error message** Diamond number for inner side **Cause of error** **Error correction**
905-27A0	**Error message** Type of dressing **Cause of error** **Error correction**
905-27A1	**Error message** Dressing strategy **Cause of error** **Error correction**

Error number	Description
905-27A2	**Error message** Removal at diameter **Cause of error** **Error correction**
905-27A3	**Error message** Removal at outer edge **Cause of error** **Error correction**
905-27A4	**Error message** Removal at inner edge **Cause of error** **Error correction**
905-27A5	**Error message** Valve number **Cause of error** **Error correction**
905-27A6	**Error message** Operation code **Cause of error** **Error correction**
905-27A7	**Error message** Type of offset 0=X,1=Z,2=X/Z **Cause of error** **Error correction**
905-27A8	**Error message** Retraction type (0=X,1=Z,2=X/Z) **Cause of error** **Error correction**
905-27A9	**Error message** Infeed direction: X component **Cause of error** **Error correction**

Error number	Description
905-27AA	**Error message** Infeed direction: Z component **Cause of error** **Error correction**
905-27AB	**Error message** Retractn amnt intermed. dressing **Cause of error** **Error correction**
905-27AC	**Error message** Retraction: X component **Cause of error** **Error correction**
905-27AD	**Error message** Retraction: Z component **Cause of error** **Error correction**
905-27AE	**Error message** Retractn. in X (incl. direction) **Cause of error** **Error correction**
905-27AF	**Error message** Retractn. in Z (incl. direction) **Cause of error** **Error correction**
905-27B0	**Error message** Retraction speed (0=FMAX) **Cause of error** **Error correction**
905-27B1	**Error message** Sensor active **Cause of error** **Error correction**

Error number	Description
905-27B2	**Error message** Dimensional control active **Cause of error** **Error correction**
905-27B3	**Error message** Dimensional control valve **Cause of error** **Error correction**
905-27B4	**Error message** Initiate interm. dressing drctly **Cause of error** **Error correction**
905-27B5	**Error message** Start the reciprocation **Cause of error** **Error correction**
905-27B6	**Error message** Start preset for X **Cause of error** **Error correction**
905-27B7	**Error message** Activation of C axis **Cause of error** **Error correction**
905-27B8	**Error message** Start preset for Y **Cause of error** **Error correction**
905-27B9	**Error message** End position **Cause of error** **Error correction**

Error number	Description
905-27BA	**Error message** Start preset for Z **Cause of error** **Error correction**
905-27BB	**Error message** Feed rate for positioning in C **Cause of error** **Error correction**
905-27BC	**Error message** Start preset for R (radius) **Cause of error** **Error correction**
905-27BD	**Error message** Start preset for H (angle) **Cause of error** **Error correction**
905-27BE	**Error message** Start preset for B **Cause of error** **Error correction**
905-27BF	**Error message** Start preset for C **Cause of error** **Error correction**
905-27C0	**Error message** Start preset for C **Cause of error** **Error correction**
905-27C1	**Error message** Tool compensation **Cause of error** **Error correction**

Error number	Description
905-27C2	**Error message** Number of revolutions **Cause of error** **Error correction**
905-27C3	**Error message** Restart noncylindrical movemnt. **Cause of error** **Error correction**
905-27C4	**Error message** Stop of noncylindrical movemnt. **Cause of error** **Error correction**
905-27C5	**Error message** Do not approach with meas. sys. **Cause of error** **Error correction**
905-27C6	**Error message** Diameter Q400 **Cause of error** **Error correction**
905-27C7	**Error message** Width Q401 **Cause of error** **Error correction**
905-27C8	**Error message** Overhang Q402 **Cause of error** **Error correction**
905-27C9	**Error message** Depth Q403 **Cause of error** **Error correction**

Error number	Description
905-27CA	**Error message** Angle Q404 **Cause of error** **Error correction**
905-27CB	**Error message** Angle Q405 **Cause of error** **Error correction**
905-27CC	**Error message** Radius Q406 **Cause of error** **Error correction**
905-27CD	**Error message** Radius Q407 **Cause of error** **Error correction**
905-27CE	**Error message** Radius Q408 **Cause of error** **Error correction**
905-27CF	**Error message** Minimum diameter **Cause of error** **Error correction**
905-27D0	**Error message** Minimum width **Cause of error** **Error correction**
905-27D1	**Error message** Wheel type **Cause of error** **Error correction**

Error number	Description
905-27D2	**Error message** Type of machining **Cause of error** **Error correction**
905-27D3	**Error message** Tool number Tool number **Cause of error** **Error correction**
905-27D4	**Error message** Angle Q414 **Cause of error** **Error correction**
905-27D5	**Error message** Cutting speed **Cause of error** **Error correction**
905-27D6	**Error message** Location number **Cause of error** **Error correction**
905-27D7	**Error message** Maximum diameter **Cause of error** **Error correction**
905-27D8	**Error message** Minimum diameter **Cause of error** **Error correction**
905-27D9	**Error message** Maximum width **Cause of error** **Error correction**

Error number	Description
905-27DA	**Error message** Minimum width **Cause of error** **Error correction**
905-27DB	**Error message** Chamfer width Q421 **Cause of error** **Error correction**
905-27DC	**Error message** Chamfer angle Q422 **Cause of error** **Error correction**
905-27DD	**Error message** Corner radius Q423 **Cause of error** **Error correction**
905-27DE	**Error message** Side length Q424 **Cause of error** **Error correction**
905-27DF	**Error message** Angle of relief Q425 **Cause of error** **Error correction**
905-27E0	**Error message** Depth of relief Q426 **Cause of error** **Error correction**
905-27E1	**Error message** Length of recess Q427 **Cause of error** **Error correction**

Error number	Description
905-27E2	**Error message** Angle of departure Q428 **Cause of error** **Error correction**
905-27E3	**Error message** Total depth Q429 **Cause of error** **Error correction**
905-27E4	**Error message** Safety clearance X **Cause of error** **Error correction**
905-27E5	**Error message** Safety clearance Z **Cause of error** **Error correction**
905-27E6	**Error message** Safety clearance Z **Cause of error** **Error correction**
905-27E7	**Error message** X edge Q433 **Cause of error** **Error correction**
905-27E8	**Error message** Minimum (V) **Cause of error** **Error correction**
905-27E9	**Error message** Maximum (V) **Cause of error** **Error correction**

Error number	Description
905-27EA	**Error message** Motor pulley **Cause of error** **Error correction**
905-27EB	**Error message** Wheel pulley **Cause of error** **Error correction**
905-27EC	**Error message** Radius Q438 **Cause of error** **Error correction**
905-27ED	**Error message** Length Q439 **Cause of error** **Error correction**
905-27EE	**Error message** Length Q440 **Cause of error** **Error correction**
905-27EF	**Error message** Location number Q441 **Cause of error** **Error correction**
905-27F0	**Error message** Logic location Q442 **Cause of error** **Error correction**
905-27F1	**Error message** Alignment Q443 **Cause of error** **Error correction**

Error number	Description
905-27F2	**Error message** Type of dresser Q444 **Cause of error** **Error correction**
905-27F3	**Error message** Actual position X Q445 **Cause of error** **Error correction**
905-27F4	**Error message** Actual position Z Q446 **Cause of error** **Error correction**
905-27F5	**Error message** Rotational speed Q447 **Cause of error** **Error correction**
905-27F6	**Error message** Width Q448 **Cause of error** **Error correction**
905-27F7	**Error message** Compensation Q449 **Cause of error** **Error correction**
905-27F8	**Error message** Nominal position X Q450 **Cause of error** **Error correction**
905-27F9	**Error message** Nominal position Z Q451 **Cause of error** **Error correction**

Error number	Description
905-27FA	**Error message** Tool type (select by soft key) **Cause of error** **Error correction**
905-27FB	**Error message** Radius Q453 **Cause of error** **Error correction**
905-27FC	**Error message** Length Q454 **Cause of error** **Error correction**
905-27FD	**Error message** Length Q455 **Cause of error** **Error correction**
905-27FE	**Error message** Diameter X Q456 **Cause of error** **Error correction**
905-27FF	**Error message** Length Z Q457 **Cause of error** **Error correction**
905-2800	**Error message** Actual position X Q458 **Cause of error** **Error correction**
905-2801	**Error message** Actual position Z Q459 **Cause of error** **Error correction**

Error number	Description
905-2802	**Error message** Select the edge Q460 **Cause of error** **Error correction**
905-2803	**Error message** Select the wheel shape Q461 **Cause of error** **Error correction**
905-2804	**Error message** Position X Q462 **Cause of error** **Error correction**
905-2805	**Error message** Position Z Q463 **Cause of error** **Error correction**
905-2806	**Error message** Number (1...4) of position Q464 **Cause of error** **Error correction**
905-2807	**Error message** Number of safety positions Q465 **Cause of error** **Error correction**
905-2808	**Error message** Datum shift X Q466 **Cause of error** **Error correction**
905-2809	**Error message** Datum shift Z Q467 **Cause of error** **Error correction**

Error number	Description
905-280A	**Error message** Select the preset Q468 **Cause of error** **Error correction**
905-280B	**Error message** Delete entry or status **Cause of error** **Error correction**
905-280C	**Error message** Measured value in axis system **Cause of error** **Error correction**
905-280D	**Error message** Axis **Cause of error** **Error correction**
905-280E	**Error message** Axis number **Cause of error** **Error correction**
905-280F	**Error message** Parameter block **Cause of error** **Error correction**
905-2810	**Error message** Probe input X12 or X13 **Cause of error** **Error correction**
905-2811	**Error message** Name of noncylindrical program **Cause of error** **Error correction**

Error number	Description
905-2812	**Error message** Name of wheel **Cause of error** **Error correction**
905-2813	**Error message** Tolerance value 1? **Cause of error** **Error correction**
905-2814	**Error message** Tolerance value 2? **Cause of error** **Error correction**
905-2815	**Error message** Feed rate limit? **Cause of error** **Error correction**
905-2816	**Error message** Ls **Cause of error** **Error correction**
905-2817	**Error message** I **Cause of error** **Error correction**
905-2818	**Error message** P **Cause of error** **Error correction**
905-2819	**Error message** T **Cause of error** **Error correction**

Error number	Description
905-281A	**Error message** D **Cause of error** **Error correction**
905-281B	**Error message** B **Cause of error** **Error correction**
905-281C	**Error message** H **Cause of error** **Error correction**
905-281D	**Error message** K **Cause of error** **Error correction**
905-281E	**Error message** C **Cause of error** **Error correction**
905-281F	**Error message** S **Cause of error** **Error correction**
905-2820	**Error message** V **Cause of error** **Error correction**
905-2821	**Error message** S **Cause of error** **Error correction**

Error number	Description
905-2822	**Error message** W **Cause of error** **Error correction**
905-2823	**Error message** L L **Cause of error** **Error correction**
905-2824	**Error message** Q **Cause of error** **Error correction**
905-2825	**Error message** X **Cause of error** **Error correction**
905-2826	**Error message** R **Cause of error** **Error correction**
905-2827	**Error message** A **Cause of error** **Error correction**
905-2828	**Error message** E **Cause of error** **Error correction**
905-2829	**Error message** Ve **Cause of error** **Error correction**

Error number	Description
905-282A	**Error message** M **Cause of error** **Error correction**
905-282B	**Error message** Vm **Cause of error** **Error correction**
905-282C	**Error message** K **Cause of error** **Error correction**
905-282D	**Error message** Vk **Cause of error** **Error correction**
905-282E	**Error message** X **Cause of error** **Error correction**
905-282F	**Error message** Z **Cause of error** **Error correction**
905-2830	**Error message** F **Cause of error** **Error correction**
905-2831	**Error message** D **Cause of error** **Error correction**

Error number	Description
905-2832	**Error message** H **Cause of error** **Error correction**
905-2833	**Error message** Tool number **Cause of error** **Error correction**
905-2834	**Error message** Error number **Cause of error** **Error correction**
905-2835	**Error message** Error consequence **Cause of error** **Error correction**
905-2836	**Error message** Error level **Cause of error** **Error correction**
905-2837	**Error message** W_PL **Cause of error** **Error correction**
905-2838	**Error message** D_PL **Cause of error** **Error correction**
905-2839	**Error message** E **Cause of error** **Error correction**

Error number	Description
905-283A	**Error message** B **Cause of error** **Error correction**
905-283B	**Error message** E **Cause of error** **Error correction**
905-283C	**Error message** NR **Cause of error** **Error correction**
905-283D	**Error message** D_Nr **Cause of error** **Error correction**
905-283E	**Error message** D1 **Cause of error** **Error correction**
905-283F	**Error message** D2 **Cause of error** **Error correction**
905-2840	**Error message** D3 **Cause of error** **Error correction**
905-2841	**Error message** K **Cause of error** **Error correction**

Error number	Description
905-2842	**Error message** Q **Cause of error** **Error correction**
905-2843	**Error message** L **Cause of error** **Error correction**
905-2844	**Error message** D **Cause of error** **Error correction**
905-2845	**Error message** Delete Z **Cause of error** **Error correction**
905-2846	**Error message** Delete X **Cause of error** **Error correction**
905-2848	**Error message** Teach in width **Cause of error** **Error correction**
905-2849	**Error message** Define the tool **Cause of error** **Error correction**
905-284A	**Error message** Continue **Cause of error** **Error correction**

Error number	Description
905-284F	**Error message** Spindle dresser **Cause of error** **Error correction**
905-2850	**Error message** Dress plate **Cause of error** **Error correction**
905-2851	**Error message** Diamond dresser **Cause of error** **Error correction**
905-2852	**Error message** Alignment **Cause of error** **Error correction**
905-2853	**Error message** Teach in dresser pos **Cause of error** **Error correction**
905-2854	**Error message** New dresser **Cause of error** **Error correction**
905-2856	**Error message** Ratio **Cause of error** **Error correction**
905-2859	**Error message** Reload the data **Cause of error** **Error correction**

Error number	Description
905-285D	**Error message** Search criteria **Cause of error** **Error correction**
905-285E	**Error message** Confirm the data **Cause of error** **Error correction**
905-2861	**Error message** Next identical **Cause of error** **Error correction**
905-2862	**Error message** Previous identical **Cause of error** **Error correction**
905-2863	**Error message** Tool information **Cause of error** **Error correction**
905-2864	**Error message** Internal grinding **Cause of error** **Error correction**
905-2865	**Error message** External grinding **Cause of error** **Error correction**
905-2866	**Error message** General wheel data **Cause of error** **Error correction**

Error number	Description
905-2867	**Error message** Face plate **Cause of error** **Error correction**
905-2868	**Error message** Angular wheel **Cause of error** **Error correction**
905-2869	**Error message** Straight wheel **Cause of error** **Error correction**
905-286A	**Error message** End **Cause of error** **Error correction**
905-286B	**Error message** X **Cause of error** **Error correction**
905-286C	**Error message** Z **Cause of error** **Error correction**
905-286D	**Error message** X **Cause of error** **Error correction**
905-286E	**Error message** Z **Cause of error** **Error correction**

Error number	Description
905-286F	**Error message** X **Cause of error** **Error correction**
905-2870	**Error message** Z **Cause of error** **Error correction**
905-2871	**Error message** X **Cause of error** **Error correction**
905-2872	**Error message** Z **Cause of error** **Error correction**
905-2873	**Error message** F1 **Cause of error** **Error correction**
905-2874	**Error message** F2 **Cause of error** **Error correction**
905-2875	**Error message** F **Cause of error** **Error correction**
905-2876	**Error message** D **Cause of error** **Error correction**

Error number	Description
905-2877	**Error message** Search path **Cause of error** **Error correction**
905-2879	**Error message** H **Cause of error** **Error correction**
905-287A	**Error message** H1 **Cause of error** **Error correction**
905-287B	**Error message** H2 **Cause of error** **Error correction**
905-287C	**Error message** N **Cause of error** **Error correction**
905-287D	**Error message** O **Cause of error** **Error correction**
905-287E	**Error message** A **Cause of error** **Error correction**
905-287F	**Error message** I **Cause of error** **Error correction**

Error number	Description
905-2880	**Error message** M91 **Cause of error** **Error correction**
905-2881	**Error message** E **Cause of error** **Error correction**
905-2882	**Error message** ER1 **Cause of error** **Error correction**
905-2883	**Error message** ER2 **Cause of error** **Error correction**
905-2884	**Error message** K **Cause of error** **Error correction**
905-2885	**Error message** ER3 **Cause of error** **Error correction**
905-2886	**Error message** ER4 **Cause of error** **Error correction**
905-2887	**Error message** ER5 **Cause of error** **Error correction**

Error number	Description
905-2888	**Error message** ER6 **Cause of error** **Error correction**
905-2889	**Error message** E **Cause of error** **Error correction**
905-288A	**Error message** E **Cause of error** **Error correction**
905-288B	**Error message** S **Cause of error** **Error correction**
905-288C	**Error message** E **Cause of error** **Error correction**
905-288D	**Error message** K **Cause of error** **Error correction**
905-288E	**Error message** E **Cause of error** **Error correction**
905-288F	**Error message** R **Cause of error** **Error correction**

Error number	Description
905-2890	**Error message** S **Cause of error** **Error correction**
905-2891	**Error message** R1 **Cause of error** **Error correction**
905-2892	**Error message** R2 **Cause of error** **Error correction**
905-2893	**Error message** R3 **Cause of error** **Error correction**
905-2894	**Error message** D **Cause of error** **Error correction**
905-2895	**Error message** F **Cause of error** **Error correction**
905-2896	**Error message** E **Cause of error** **Error correction**
905-2897	**Error message** H **Cause of error** **Error correction**

Error number	Description
905-2898	**Error message** Q **Cause of error** **Error correction**
905-2899	**Error message** N **Cause of error** **Error correction**
905-289A	**Error message** Diamond no. outside **Cause of error** **Error correction**
905-289B	**Error message** Diamond no. inside **Cause of error** **Error correction**
905-289C	**Error message** Type of dressing **Cause of error** **Error correction**
905-289D	**Error message** Dressing strategy **Cause of error** **Error correction**
905-289E	**Error message** D **Cause of error** **Error correction**
905-289F	**Error message** O **Cause of error** **Error correction**

Error number	Description
905-28A0	**Error message** I **Cause of error** **Error correction**
905-28A1	**Error message** V **Cause of error** **Error correction**
905-28A2	**Error message** C **Cause of error** **Error correction**
905-28A3	**Error message** RL **Cause of error** **Error correction**
905-28A4	**Error message** RA **Cause of error** **Error correction**
905-28A5	**Error message** dX **Cause of error** **Error correction**
905-28A6	**Error message** dZ **Cause of error** **Error correction**
905-28A7	**Error message** AZ **Cause of error** **Error correction**

Error number	Description
905-28A8	**Error message** dXA **Cause of error** **Error correction**
905-28A9	**Error message** dZA **Cause of error** **Error correction**
905-28AA	**Error message** AX **Cause of error** **Error correction**
905-28AB	**Error message** AZ **Cause of error** **Error correction**
905-28AC	**Error message** FA **Cause of error** **Error correction**
905-28AD	**Error message** SA **Cause of error** **Error correction**
905-28AE	**Error message** MA **Cause of error** **Error correction**
905-28AF	**Error message** MV **Cause of error** **Error correction**

Error number	Description
905-28B0	**Error message** D_OK **Cause of error** **Error correction**
905-28B1	**Error message** P **Cause of error** **Error correction**
905-28B2	**Error message** X **Cause of error** **Error correction**
905-28B3	**Error message** E **Cause of error** **Error correction**
905-28B4	**Error message** Y **Cause of error** **Error correction**
905-28B5	**Error message** C **Cause of error** **Error correction**
905-28B6	**Error message** Z **Cause of error** **Error correction**
905-28B7	**Error message** F **Cause of error** **Error correction**

Error number	Description
905-28B8	**Error message** R **Cause of error** **Error correction**
905-28B9	**Error message** H **Cause of error** **Error correction**
905-28BA	**Error message** B **Cause of error** **Error correction**
905-28BB	**Error message** C **Cause of error** **Error correction**
905-28BC	**Error message** C **Cause of error** **Error correction**
905-28BD	**Error message** COR **Cause of error** **Error correction**
905-28BE	**Error message** COUNT **Cause of error** **Error correction**
905-28BF	**Error message** RESET **Cause of error** **Error correction**

Error number	Description
905-28C0	**Error message** STOPP **Cause of error** **Error correction**
905-28C1	**Error message** U **Cause of error** **Error correction**
905-28C2	**Error message** Diameter **Cause of error** **Error correction**
905-28C3	**Error message** Width of wheel **Cause of error** **Error correction**
905-28C4	**Error message** Overhang **Cause of error** **Error correction**
905-28C5	**Error message** Depth of wheel **Cause of error** **Error correction**
905-28C6	**Error message** Wheel tilt angle **Cause of error** **Error correction**
905-28C7	**Error message** Corner angle **Cause of error** **Error correction**

Error number	Description
905-28C8	**Error message** Corner radius RV **Cause of error** **Error correction**
905-28C9	**Error message** Corner radius RV1 **Cause of error** **Error correction**
905-28CA	**Error message** Corner radius RV2 **Cause of error** **Error correction**
905-28CB	**Error message** Minimum diameter **Cause of error** **Error correction**
905-28CC	**Error message** Minimum width **Cause of error** **Error correction**
905-28CD	**Error message** Straight,angul.,flat **Cause of error** **Error correction**
905-28CE	**Error message** External / Internal **Cause of error** **Error correction**
905-28D0	**Error message** Offset B axis **Cause of error** **Error correction**

Error number	Description
905-28D1	**Error message** Cutting speed **Cause of error** **Error correction**
905-28D2	**Error message** Location no. (0..99) **Cause of error** **Error correction**
905-28D3	**Error message** Upper limit diameter **Cause of error** **Error correction**
905-28D4	**Error message** Lower limit diameter **Cause of error** **Error correction**
905-28D5	**Error message** Upper limit width **Cause of error** **Error correction**
905-28D6	**Error message** Lower limit width **Cause of error** **Error correction**
905-28D7	**Error message** Width **Cause of error** **Error correction**
905-28D8	**Error message** Angle **Cause of error** **Error correction**

Error number	Description
905-28D9	**Error message** Radius **Cause of error** **Error correction**
905-28DA	**Error message** Side length **Cause of error** **Error correction**
905-28DB	**Error message** Angle **Cause of error** **Error correction**
905-28DC	**Error message** Depth **Cause of error** **Error correction**
905-28DD	**Error message** Length **Cause of error** **Error correction**
905-28DE	**Error message** Radius **Cause of error** **Error correction**
905-28DF	**Error message** Depth **Cause of error** **Error correction**
905-28E0	**Error message** Diameter **Cause of error** **Error correction**

Error number	Description
905-28E1	**Error message** Outer side **Cause of error** **Error correction**
905-28E2	**Error message** Inner side **Cause of error** **Error correction**
905-28E3	**Error message** Edge selection **Cause of error** **Error correction**
905-28E4	**Error message** Cutting speed **Cause of error** **Error correction**
905-28E5	**Error message** Cutting speed **Cause of error** **Error correction**
905-28E6	**Error message** Diameter **Cause of error** **Error correction**
905-28E7	**Error message** Diameter **Cause of error** **Error correction**
905-28E8	**Error message** Diamond radius **Cause of error** **Error correction**

Error number	Description
905-28E9	**Error message** Length L1 **Cause of error** **Error correction**
905-28EA	**Error message** Length L2 **Cause of error** **Error correction**
905-28EB	**Error message** Dresser location **Cause of error** **Error correction**
905-28EC	**Error message** Dresser location **Cause of error** **Error correction**
905-28ED	**Error message** Alignment **Cause of error** **Error correction**
905-28EE	**Error message** Type of dresser **Cause of error** **Error correction**
905-28EF	**Error message** Dresser position **Cause of error** **Error correction**
905-28F0	**Error message** Dresser position **Cause of error** **Error correction**

Error number	Description
905-28F1	**Error message** Dressing spindle **Cause of error** **Error correction**
905-28F2	**Error message** Width of dress plate **Cause of error** **Error correction**
905-28F3	**Error message** Type of compensation **Cause of error** **Error correction**
905-28F4	**Error message** Dresser position **Cause of error** **Error correction**
905-28F5	**Error message** Dresser position **Cause of error** **Error correction**
905-28F6	**Error message** Tool type **Cause of error** **Error correction**
905-28F7	**Error message** Probe-tip radius **Cause of error** **Error correction**
905-28F8	**Error message** Length L1 **Cause of error** **Error correction**

Error number	Description
905-28F9	**Error message** Length L2 **Cause of error** **Error correction**
905-28FA	**Error message** Calibration **Cause of error** **Error correction**
905-28FF	**Error message** Wheel shape **Cause of error** **Error correction**
905-2903	**Error message** Safety position. Safety position. Safety position. Safety position. **Cause of error** **Error correction**
905-2905	**Error message** Datum shift Datum shift **Cause of error** **Error correction**
905-2906	**Error message** Calibration Calibration Calibration Calibration Calibration **Cause of error** **Error correction**
905-2907	**Error message** E **Cause of error** **Error correction**

Error number	Description
905-2908	**Error message** Axis system **Cause of error** **Error correction**
905-2909	**Error message** Radius **Cause of error** **Error correction**
905-290A	**Error message** A **Cause of error** **Error correction**
905-290B	**Error message** Axis **Cause of error** **Error correction**
905-290C	**Error message** P **Cause of error** **Error correction**
905-290D	**Error message** Probe input **Cause of error** **Error correction**
905-290E	**Error message** PGM_NAME **Cause of error** **Error correction**
905-290F	**Error message** Wheel name **Cause of error** **Error correction**

Error number	Description
905-2910	**Error message** TOLERANCE 1 **Cause of error** **Error correction**
905-2911	**Error message** TOLERANCE 2 **Cause of error** **Error correction**
905-2912	**Error message** Feed rate limit **Cause of error** **Error correction**
905-2913	**Error message** Wheel data Wheel data Wheel data Wheel data **Cause of error** **Error correction**
905-2917	**Error message** Outside **Cause of error** **Error correction**
905-2918	**Error message** Inside **Cause of error** **Error correction**
905-2919	**Error message** Settings **Cause of error** **Error correction**
905-291A	**Error message** Cutting speed Cutting speed **Cause of error** **Error correction**

Error number	Description
905-291C	**Error message** Ratio **Cause of error** **Error correction**
905-291D	**Error message** Insertion location 0 **Cause of error** **Error correction**
905-291E	**Error message** Insertion location 1 **Cause of error** **Error correction**
905-291F	**Error message** Insertion location 2 **Cause of error** **Error correction**
905-2920	**Error message** Insertion location 3 **Cause of error** **Error correction**
905-2921	**Error message** Insertion location 4 **Cause of error** **Error correction**
905-2922	**Error message** Insertion location 5 **Cause of error** **Error correction**
905-2923	**Error message** Insertion location 6 **Cause of error** **Error correction**

Error number	Description
905-2924	**Error message** Insertion location 7 **Cause of error** **Error correction**
905-2925	**Error message** Insertion location 8 **Cause of error** **Error correction**
905-2926	**Error message** Insertion location 9 **Cause of error** **Error correction**
905-2927	**Error message** Wheel data **Cause of error** **Error correction**
905-2928	**Error message** Cutter data **Cause of error** **Error correction**
905-2929	**Error message** Dresser data **Cause of error** **Error correction**
905-292A	**Error message** Probe data **Cause of error** **Error correction**
905-292B	**Error message** Drill data **Cause of error** **Error correction**

Error number	Description
905-292C	**Error message** Search criteria **Cause of error** **Error correction**
905-292D	**Error message** Probe **Cause of error** **Error correction**
905-292E	**Error message** New dresser **Cause of error** **Error correction**
905-292F	**Error message** Define dresser Define dresser Define dresser **Cause of error** **Error correction**
905-2932	**Error message** Define alignment **Cause of error** **Error correction**
905-2933	**Error message** Dresser position Dresser position **Cause of error** **Error correction**
905-2935	**Error message** Calibration: Select **Cause of error** **Error correction**
905-2936	**Error message** Teach in positions **Cause of error** **Error correction**

Error number	Description
905-2937	**Error message** Teach in width **Cause of error** **Error correction**
905-2938	**Error message** Wheel form **Cause of error** **Error correction**
905-2939	**Error message** Wrkpc. datum shift **Cause of error** **Error correction**
905-293A	**Error message** Wheel datum shift **Cause of error** **Error correction**
905-293B	**Error message** Safety position **Cause of error** **Error correction**
905-293C	**Error message** No. of safety pos. **Cause of error** **Error correction**
905-293D	**Error message** T command **Cause of error** **Error correction**
905-293E	**Error message** Q command **Cause of error** **Error correction**

Error number	Description
905-2940	**Error message** Thread grinding **Cause of error** **Error correction**
905-2941	**Error message** Thread plunging **Cause of error** **Error correction**
905-2942	**Error message** Thread oscillation **Cause of error** **Error correction**
905-2943	**Error message** C axis **Cause of error** **Error correction**
905-2944	**Error message** Start of grinding **Cause of error** **Error correction**
905-2945	**Error message** End of grinding **Cause of error** **Error correction**
905-2946	**Error message** Reciprocation config **Cause of error** **Error correction**
905-2947	**Error message** Recipr.para. config. **Cause of error** **Error correction**

Error number	Description
905-2948	**Error message** Start pos. infeed **Cause of error** **Error correction**
905-2949	**Error message** Asynchronous infeed **Cause of error** **Error correction**
905-294A	**Error message** Synchronous infeed **Cause of error** **Error correction**
905-294B	**Error message** General infeed **Cause of error** **Error correction**
905-294C	**Error message** Infeed meas.sys.asyn **Cause of error** **Error correction**
905-294D	**Error message** Infeed meas.sys.sync **Cause of error** **Error correction**
905-294E	**Error message** Probe infeed **Cause of error** **Error correction**
905-294F	**Error message** Infeed gap control **Cause of error** **Error correction**

Error number	Description
905-2950	**Error message** Infeed probe **Cause of error** **Error correction**
905-2951	**Error message** General dressing **Cause of error** **Error correction**
905-2952	**Error message** Intermed. dressing **Cause of error** **Error correction**
905-2953	**Error message** Load contour program **Cause of error** **Error correction**
905-2954	**Error message** Start of contour pgm **Cause of error** **Error correction**
905-2955	**Error message** End of contour pgm **Cause of error** **Error correction**
905-2956	**Error message** Start of contour pgm **Cause of error** **Error correction**
905-2957	**Error message** Contour program stop **Cause of error** **Error correction**

Error number	Description
905-2958	**Error message** Asynchr. infeed def. **Cause of error** **Error correction**
905-2959	**Error message** Synchr. infeed def. **Cause of error** **Error correction**
905-295A	**Error message** General infeed def. **Cause of error** **Error correction**
905-295B	**Error message** Override assignment **Cause of error** **Error correction**
905-295C	**Error message** Start of grinding **Cause of error** **Error correction**
905-295D	**Error message** Grinding stop **Cause of error** **Error correction**
905-295E	**Error message** Grinding status **Cause of error** **Error correction**
905-295F	**Error message** Wait f. grinding end **Cause of error** **Error correction**

Error number	Description
905-2960	**Error message** Activate event **Cause of error** **Error correction**
905-2961	**Error message** Deactivate event **Cause of error** **Error correction**
905-2962	**Error message** Check event **Cause of error** **Error correction**
905-2963	**Error message** Wheel data **Cause of error** **Error correction**
905-2964	**Error message** Wheel head **Cause of error** **Error correction**
905-2965	**Error message** Dresser data **Cause of error** **Error correction**
905-2966	**Error message** Calibration **Cause of error** **Error correction**
905-2967	**Error message** Safety position **Cause of error** **Error correction**

Error number	Description
905-2968	**Error message** Wrkpc. datum shift **Cause of error** **Error correction**
905-2969	**Error message** Wheel datum shift **Cause of error** **Error correction**
905-296A	**Error message** T command **Cause of error** **Error correction**
905-296B	**Error message** Q command **Cause of error** **Error correction**
905-296C	**Error message** Wheel status **Cause of error** **Error correction**
905-296D	**Error message** Delete the dresser **Cause of error** **Error correction**
905-296E	**Error message** Select param. block **Cause of error** **Error correction**
905-296F	**Error message** Start dressing **Cause of error** **Error correction**

Error number	Description
905-2970	**Error message** Reverse dressing **Cause of error** **Error correction**
905-2971	**Error message** Compensate wheel **Cause of error** **Error correction**
905-2972	**Error message** Reference system **Cause of error** **Error correction**
905-2973	**Error message** PLC command **Cause of error** **Error correction**

Index

T

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
E-mail: service.app@heidenhain.de

www.heidenhain.com

Original document
1234567 · 00 · A · 01 · 01/2022 · PDF